NOV CTES



Cerberus User Guide

Coiled Tubing, Jointed Pipe, Wireline, and Slickline

Version 11.0

March 2013

Copyright

Copyright © 1997-2013 NOV CTES

All rights reserved.

While every precaution has been taken in the preparation of this document, the publisher assumes no responsibility for errors or omissions. Information in this documentation is subject to change without notice. No liability is assumed for damages resulting from the use of the information contained herein.

About Cerberus

When running tools into and out of a well on coiled tubing, wireline, slickline, or pipe, operations use Cerberus modeling software to accurately predict and analyze cumulative forces and fatigue at each stage of a job. Cerberus is able to determine whether the target depth can be reached, the desired tasks performed, and the equipment safely returned to surface.

The application's most important feature is its ability to model conditions in deviated and horizontal wellbores. Cerberus is the only commercial program able to model all three conveyance methods in one package, making it invaluable for choosing the best well-entry technique in marginal conditions.

Contact Us

United States

NOV Corporate Headquarters
7909 Parkwood Circle Drive
Houston, Texas 77036
Phone: 1 (713) 375-3700
Fax: 1 (713) 346-7687

NOV CTES
3770 Pollok Drive
Conroe, Texas 77303
Phone: 1 (936) 777 6200
Fax: 1 (936) 777 6312
After hours technical support: 1 (936) 777 6200

email:CTESSales@nov.com

OR

CTESSupport@nov.com

Website:http://www.nov.com/ctes

United Kingdom

Unit 16
Enterprise Drive
Westhill Industrial Estate
Westhill
Aberdeen
AB32 6TQ
Phone: 44 1224 748797
Fax: 44 1224 741129

Middle East

B-17, Oilfield Supply Center
Jebel Ali Free Zone
P.O. Box 18633
Dubai
United Arab Emirates
Phone: +971 481 100100
Fax: +971 488 37087

Contents, Cont'd.

Copyright ii
About Cerberus iii
Contact Us iv
United States iv
United Kingdom iv
Middle East iv

Chapter 1
Installing or Updating Cerberus 1
System Requirements 2
Basic Requirements 2
Downloading Cerberus from the Internet 2
Downloading Updates and/or Patches from the Internet 2
Running Setup 3
Starting Cerberus and Obtaining Authorization 4
Moving Cerberus Data to a Network 5
Backing Up and Restoring Data Files 5
Moving Cerberus' Data Directory (First Computer) 6
Changing Cerberus' Data Directory Location (Other Computers) 6
Troubleshooting Installation 7

Chapter 2
System Setup 9
Using the System Setup Options 10
Managing Units 10
Unit Sets 10
Individual Units 11
Changing Authorization 12
Using Cerberus Setup 14
Viewing Records Checked Out 16
Undoing a Record Check-out 16
Copying a Data Directory 17
Changing a Data Directory 17

Chapter 3
Using Cerberus 19
Getting Started 20
Starting Cerberus 21
The Main Menu (Home Page) 21

Contents, Cont'd.

The Main-menu Layout 22

Chapter 4

Reel-Trak 29

About Coiled Tubing (CT) Fatigue Tracking with Reel-Trak 30
Creating a New Job 30
Using the Job Wizard 31
Using the New Job Dialog 33
Setting Up a CT Job 34
Tracking Fatigue 42
Setting Up Fatigue Tracking Display Options 43
Understanding the Fatigue Diagram 45
Understanding the Position Locator 45
Understanding the String Diagram 45
Running the Fatigue Job 46
Track Fatigue Toolbar 46
Running the Job from Recorded Data 47
Running the Job in Real Time 49
Running the Job in Playback Mode 49
Using the Post Job Analysis Tab 50
Using the Job Log 53
Setting Up Job Log Display Options 53
Job-log Toolbar Buttons 54
Job-log Menu-bar Selections 55
Packaging a Job 58

Chapter 5

Using Orpheus 61

About Orpheus 62
The Orpheus Process 62
Accessing Orpheus 63
Working with Projects 64
Creating a New Orpheus Project 64
Opening an Existing Orpheus Project 67
Organizing Projects into Folders 67
Saving an Orpheus Project 67
Deleting an Orpheus Project 68
Importing an Orpheus Project 68

Contents, Cont'd.

Exporting an Orpheus Project 68
Entering Project-configuration Information 69
Entering Project Options 77
Setting Advanced Options/Calculation Parameters 80
Setting Project-option Defaults 84
Performing Calculations 85
Calculation Methods 85
Standard Calculations 85
Analysis Tools 103
Printing Orpheus Project Data 131
Emailing Project Data 132

Chapter 6
Using Hydra 135
About Hydra 136
The Hydra Process 136
Accessing Hydra 138
Hydra's Opening-dialog Tools 139
Working with Projects 141
Creating a New Hydra Project 141
Opening an Existing Hydra Project 143
Organizing Projects into Folders 144
Saving a Hydra Project 144
Deleting a Hydra Project 145
Importing a Hydra Project 145
Exporting a Hydra Project 145
Entering Project-configuration Information 145
Performing Calculations 151
Standard "Snapshot" Calculations 152
Stage-table Calculations 154
Application-wizard Calculations 159

Chapter 7
Using Velocity String 169
The Velocity String Process 170
Accessing Velocity String (Hydra) 171
Working with Projects 171
Using the Velocity String Opening-dialog Tools 172
Setting Program Preferences 172
Creating a New Velocity String Project 172

Contents, Cont'd.

Opening an Existing Velocity String Project 173
Saving a Velocity String Project 174
Deleting a Velocity String Project 174
Importing a Velocity String Project 175
Exporting a Velocity String Project 175
Entering Project-configuration Information 175
Select a Well with a Defined Reservoir 175
Verify Reservoir Parameters 176
Select One or More Velocity Strings 177
Enter Runtime Data 179
Calculating Data 179
Using the Output Data 180

Chapter 8
Using the Packer and Completion Analysis (PACA) 183
About PACA 184
The PACA Process 185
Accessing the PACA Model/Wizard 186
Setting Program Preferences 186
Accessing the Well Manager and Fluids Manager 186
Working with Projects 186
Creating a New PACA Project 186
Opening an Existing PACA Project 187
Saving a PACA Project 188
Deleting a PACA Project 188
Importing a PACA Project 189
Exporting a PACA Project 189
Using the PACA Wizard 189
Entering Calculation Options 189
Selecting the PACA Application 189
Following the "Permanent Tubing Completion" Path 190
Selecting a Well that Includes Tubing String and Packer 190
Following the "Temporary Packer Run on Coiled Tubing or Pipe" Path ... 191
Selecting a Well 191
Selecting the Coiled Tubing String or Jointed Pipe 192
Selecting a Tool with a Packer 192
Following the Converged Path 194
Configuring Packer Properties 194
Defining Initial Conditions 195

Contents, Cont'd.

Defining One or More Subsequent Scenarios 195
Conducting the Analysis 196

Chapter 9
Using Solids Cleanout 199
The Solids Cleanout Process 200
Accessing Solids Cleanout (Hydra) 200
Working with Projects 201
Using the Solids Cleanout Opening-dialog Tools 201
Setting Program Preferences 202
Creating a New Solids Cleanout Project 202
Opening an Existing Solids Cleanout Project 203
Saving a Solids Cleanout Project 204
Deleting a Solids Cleanout Project 204
Importing a Solids Cleanout Project 204
Exporting a Solids Cleanout Project 205
Entering Project-configuration Information 205
Select a Well 205
Enter Fill Data 206
Select Equipment (Reel, Coiled Tubing, and Tool String) 206
Select Fluid Data 208
Enter Constraints 209
Calculate Sensitivity 210
Enter Pump Rates for Each Stage 211
Adjust Any Other Operating Parameters 212
Perform the Job-design Calculation 213
Using the Output Data 214

Chapter 10
Using the String Editor 215
Starting String Editor 215
The String Editor Display Screen 216
String Editor Toolbar 216
String Summary 218
Properties 218
Info 219
String Diagram 220
Working with Strings 220
Creating a New String 220
Manually Creating a New String 220

Contents, Cont'd.

Creating a String using the String Configuration Wizard 221
Creating a String using the String Design Wizard 222
Default String Options 228
Opening an Existing String 228
Importing a String 229
Exporting a String 229
Saving a String 230
Saving a String with or without Historical Data..... 231
Saving a Copy of a String 231
Deleting a String 231
Using CT Specifications 232
Locking or Unlocking a String 232
Archiving a String 233
Printing String Data 234
Printing a Single String Report..... 234
Printing a Zoomed View of the String Life..... 236
Receiving a String on Removable Media from a Manufacturer 237
Using CT Specifications 237
Configuring Strings 238
Cerberus's View of a String 238
What is Segment Length?..... 239
What is Position on the String? 239
What is Derating? 239
Required String Data 240
Configuring Existing Fatigue on a Used String 240
Configuring Diameter 240
Configuring Length 241
Configuring Date Commissioned 241
Configuring Sections 242
What are Sections? 242
Configuring Sections by Wall Size 243
Configuring Sections by Strip 244
Notes on Wall Reduction..... 245
Configuring Welds 246
Adding a Weld..... 246
Editing a Weld..... 247
Deleting a Weld..... 247
Configuring a Cable in String Editor 247
Configuring Zones 248

Contents, Cont'd.

What are Zones? 248
Adding a Zone 249
Removing a Zone 249
Configuring Corrosion 250
About the Suggested Corrosion Derating Factors 250
Entering Corrosion Settings for a String 250
String Utilities 251
String Weight 251
String Volume 252
Estimating Remaining Trips 252
String Services 254
Reversing a String 254
Cutting a String 255
Cutting Off the End of a String 255
Cutting a String in the Middle 256
Cutting a String and Saving Both Parts as Separate Strings 256
Splicing Strings 258
Compatibility Requirements for Splicing Strings 258
Adding New Pipe to the End of a String 258
Adding New Pipe to the Middle of a String 260
Splicing Strings End to End 261
Splicing a String into the Middle of Another String 262
String History 263
Undoing String Events 264
String Fatigue 264
Troubleshooting in String Editor 267

Chapter 11

Using the Tool String Editor 269

Starting the Tool String Editor 270
Navigating the Tool String Editor 270
Toolbox Area 271
ToolString Area 272
Working with Toolboxes 273
Creating a New Toolbox 274
Creating a New Tool Inside a Toolbox 274
Editing a Tool Within a Toolbox 275
Deleting a Tool from a Toolbox 276
Copying an Existing Tool Into a Tool Box 276
Finding Tools by Grouping 276

Contents, Cont'd.

Finding Tools using Tool Search 277
Working with Tool Strings 278
Adding Tools to a Tool String 280
Moving Tools within a Tool String 281
Removing Tools from a Tool String 282
Editing a Tool within a Tool String 284
Working with the Tool String Tab 284
Printing a Tool String 285

Chapter 12
Using the Tool Editor 287
About the Tool Editor 288
Launching the Tool Editor 288
Setting Tool Properties 288
Setting Tool Features 289
Setting Advanced Tool and Section Features 290
Setting Jar Tool Features (WL) 290
Setting Normal Force Section Features (WL) 291
Setting the Tool Image 291
Defining Tool Sections 294
Defining Tool Section Features 296
Printing a Tool 299
Customizing a Tool Printout 299

Chapter 13
Using the Well Editor 301
Starting Well Editor 302
The Well Editor Display Screen 303
The Well Editor Tools 304
The Main Well Editor Toolbar 304
Viewing a Well 305
Viewing Survey Data Graphs 306
Working with Wells 307
Creating a New Well 307
Opening an Existing Well 308
Saving a Well 308
Deleting a Well 308
Importing a Well 309

Contents, Cont'd.

Exporting a Well 309
Printing Well Data 310
Printing a Custom View of the Survey Path 311
Emailing Well Data Files 312
Configuring Wells 312
Using Quick Well 313
Configuring Depth 313
Configuring Subsea Information 314
Configuring Casing 315
Configuring Liners 315
Configuring Tubing 316
Configuring a Drill Pipe 316
Configuring the Open Hole 317
Configuring the Bridge Plug 317
Configuring the Reservoir 317
Configuring Friction 318
Configuring Survey Data 319
Entering Survey Points 320
Using the Survey Wizard 320
Adjusting Survey Tortuosity 321
Exporting Survey Data 322
Importing Survey Data 323
Configuring Temperature 324
Configure Location and Miscellaneous Information 325
Troubleshooting in Well Editor 326

Chapter 14

Using Reel Editor 327
Starting Reel Editor 327
The Reel Editor Screen 328
The Reel Editor Toolbars 328
Working with Reels 329
Creating a New Reel 330
Opening an Existing Reel 330
Saving a Reel 330
Deleting a Reel 331
Importing a Reel 332
Exporting a Reel 332
Locking a Reel 333
Unlocking a Reel 333

Contents, Cont'd.

Archiving a Reel 334
Printing Reel Data 334
Configuring Reels 335
Configuring Reel Geometry 335
Configuring Reel Type 336
Configuring Service Data 337
Estimating Reel Capacity 337
Troubleshooting in Reel Editor 338

Chapter 15
Using Fluid Manager 339
Starting Fluid Manager 339
The Fluid Manager Display Screen 339
Using the Fluid Manager Toolbar 340
Types of Fluids 340
Liquid 341
Gas 341
Foam 341
Nodal (Multiphase) Fluid 341
Newtonian Fluid 342
Bingham Plastic 342
Power Law Fluid 342
Herschel-Bulkley 343
Working with Fluids 343
Creating a New Fluid 343
Opening an Existing Fluid 343
Saving a Fluid 344
Deleting a Fluid 344
Importing a Fluid 344
Exporting a Fluid 345
Configuring a Foam 345
Calculating Properties for Liquids 346
Fluid Correction Factors 347
Entering Fluid Correction Factors 347
Using Fluid Correction Factors 347
Troubleshooting in Fluid Manager 348

Contents, Cont'd.

Chapter 16

Using Equipment Manager 349

Starting Equipment Manager 349
The Equipment Manager Screen 349
The Equipment Manager Toolbars 350
Working with Pieces of Equipment 351
Creating New Equipment 352
Opening Existing Equipment 352
Saving Equipment 352
Deleting Equipment 353
Creating an Equipment Package 353

Chapter 17

Using Personnel Manager 355

Starting Personnel Manager 355
The Personnel Manager Screen 355
The Personnel Manager Toolbars 356
Working with Personnel Records 357
Creating a New Personnel Record 357
Opening an Existing Personnel Record 357
Saving a Personnel Record 358
Deleting a Personnel Record 358

Chapter 18

Using Achilles 359

Starting Achilles 360
Fatigue Model Theory 360
Bending Events for CT 360
Fatigue Model Inputs 361
How to Minimize Fatigue 361
Using Achilles 361
The Achilles Interface 361
The Achilles Toolbar 362
Using the Inputs Tab 362
Using the Analyzer Tab 364
Using the Analyzer Data Tab 366
Using the Options Tab 367

Contents, Cont'd.

Chapter 19

Using Hercules 369
Starting Hercules 369
Tubing Limits Theory 369
Coiled Tubing Stresses 369
Axial Stress 370
Radial Stress 370
Hoop Stress 370
Torque 371
von Mises Yield Condition 371
Limit Curves 371
Maximum Diameter Considerations 371
How the Differential Pressures Curve is Calculated 371
How Constant Pressures Curves are Calculated 372
What is Imaginary Data? 372
Applying Safety Factors 372
The Hercules Display Screen 373
The Hercules Toolbar 373
Working with Hercules Projects 374
Creating a New Hercules Project 374
Opening an Existing Hercules Project 374
Saving a Hercules Project 374
Deleting a Hercules Project 375
Calculating Results in Hercules 375
Calculating Limits 375
The Differential Pressures Curve 376
Reading the Differential Pressures Graph 376
Troubleshooting in Hercules 377

Appendix A

Setting Graph Display Options in Orpheus 379
Configuring Standard Calculation Graphs 380
Calculating Yield Limit and Lockup Limit 380
Changing Default Curve-style Settings 381
Changing Curve Properties 383
Freezing a Line on a Graph 384
Hiding a Line on a Graph 386
Showing Hidden Lines on a Graph 386

Contents, Cont'd.

Appendix B
Coiled Tubing Forces Theory 389
- Sinusoidal Buckling 390
- Helical Buckling 390
- Lockup Theory 390
- Effect of Curvature on Helical Buckling Load 391
- Residual Bend and Friction Coefficients 391
- Real Force vs. Effective Force 391
- Force vs. Weight 392
- Factors Affecting Weight 392
- Capstan or Belt Effect 393
- Free Fall 393
- How to Get More Set Down Force 394
- Torque 394
- Finite Element Analysis 395

Appendix C
Advanced Fluids-model Options in Orpheus 397
- Fluids Model: Advanced Hydraulics Options 398

Appendix D
Features and Effects Options in Orpheus 401
- Modeling the Effects of the NOV Downhole Agitator Tool (CT/JP) 402
 - Including the Effect of the Agitator Tool in Calculations 402
 - Calculating Lockup Depth 403
 - Calculating Trip In and Trip Out 406
- Completing Jar Activation Calculations (WL) 408
- Including Additional Normal Force in Calculations (WL) 412

Appendix E
Suggested Readings 415
- Suggested Readings on Fatigue 416
- Suggested Readings on Forces 417
- Suggested Readings on Limits 418
- Suggested Readings on Hydraulics 418

Appendix F
Glossary 421

Chapter 1
Installing or Updating Cerberus

Before using Cerberus for the first time, install Cerberus on a Microsoft Windows®-based computer and obtain authorization.

System Requirements

The Cerberus software suite is calculations- and graphics-intensive. When considering computer choices, keep in mind having additional RAM noticeably improves performance over having a faster processor speed.

Basic Requirements

- Microsoft Windows XP, Windows Vista, Windows 7, or Windows 8 operating system
- Processor requirement: 1.5 GHz or higher; 2.4 GHz recommended
- 2 GB RAM, or more
- 200 MB free disk space
- 1024x768 screen resolution
- Mouse or other pointing device

Downloading Cerberus from the Internet

➢ To download Cerberus

1. Make sure that an Internet connection is established.
2. Point the web browser to www.nov.com/ctes. Navigate to the customer download page.
3. Enter the UserName and (case sensitive) password information received from NOV CTES.



Try to schedule downloads for times when an Internet connection can be maintained for a long period of time.

4. Download and save the software to a Windows-based computer.

Downloading Updates and/or Patches from the Internet

Use the Internet update utility to check for updates and download files.

➢ To check for and download updates over the Internet

1. Make sure that an Internet connection is established.
2. Exit all applications except Cerberus.
3. Find the **Cerberus Main Menu**.
4. From the **Help** menu, select **Check for Updates**.

 Updates can also be obtained by visiting our website: www.nov.com/ctes.

Depending the size of the file and the connection speed, updates can take a long time to download. Scheduling updates for times when an Internet connection can be maintained for a long period of time may be helpful.

Running Setup

Be prepared to obtain authorization before running the software, even on previously authorized computers.

If problems occur while installing Cerberus, see "Contact Us" on page iv.

➢ To run Setup to install or to update Cerberus

1. If running a Windows XP, Windows Vista, Windows 7, or Windows 8 operating system, log on as a user with administrator rights.
2. Close all running applications.
3. From the location in Windows where the software was saved, run `CerberusInstall.exe`.

The case-sensitive password information must be entered again.

4. The CTES installation wizard appears. Follow the instructions on the screen.



5. Obtain authorization for this copy of Cerberus (see page 4).

For Cerberus 7.0 and earlier users: If a path (for the data directory) other than the default path (c:\documents and settings\all users\application data\ctes\data) is chosen, be sure NOT to choose the same data path currently in use. Doing so will cause Cerberus to overwrite the previous data directory. This only applies to versions 7.0 and earlier.

Starting Cerberus and Obtaining Authorization

The first time Cerberus is installed authorization must be obtained from NOV CTES. The authorization contains information about the available Cerberus features and the expiration date of the software license.

➢ To start Cerberus

1. Click the **Start** button in the task bar.
2. Click **All Programs** > **CTES** > **Cerberus 11.0**.
3. When Cerberus is launched, the program prompts for authorization. Click **Yes**. The Authorization screen appears.
4. If connected to the Internet, use Option 1 to obtain authorization over the Internet. (Note that if behind a firewall, this option might be unavailable.)

 Click the **Internet** button, enter the contact information, and click **OK**. Enter information in **all** fields.
5. If not connected to the Internet, or if the Internet authorization fails, use Option 2 to obtain authorization. If possible, remain in front of the computer when contacting CTES.

 Contact NOV CTES (CTESKeys@nov.com) with an authorization code to receive an access key. When an access key is obtained from an NOV CTES employee, enter the value in the box underneath the **Authorization Code** field; click **Verify**.
6. The program's physical location cannot be moved when authorization is complete.

Authorization screen



Moving Cerberus Data to a Network

If Cerberus is being used on a stand-alone computer, the data can be moved to a network where several people can share a common set of data files. If there are data files on additional computers, they can be uploaded to the network as well.

- "Backing Up and Restoring Data Files" on page 5.
- "Moving Cerberus' Data Directory (First Computer)" on page 6.
- "Changing Cerberus' Data Directory Location (Other Computers)" on page 6.

Backing Up and Restoring Data Files

Data files can be backed up to other media, including any removable media the computer accepts. Once the data is backed up, it can also be restored in the event of data loss on the computer, such as from a hard-disk crash.

➢ To back up data files

1. Start Cerberus.
2. From the **Utilities** menu, select **Backup & Restore** > **Backup Data Directory**.
3. Choose the location for the backup file; select **Save**. When the backup is complete, click **OK**.

➢ To restore data files

1. Start Cerberus.
2. From the **Utilities** menu, select **Backup > Restore data directory...**.
3. Choose the backup file to restore, select **Open**. To confirm, click **Yes**.
4. When the data has been restored, click **OK**.



When restoring data from a backup file, all existing data files are replaced with the files from the backup. You will lose all data that is not in the backup.

Moving Cerberus' Data Directory (First Computer)

Data files can be moved from a computer to a network to allow other users to access this data.

➢ To move data from a computer to a network or other location (first computer)

1. Start Cerberus. Back up the Cerberus directory by clicking, from the **Utilities** menu, **Backup & Restore** > **Backup Data Directory**.
2. Copy the data directory from the local computer to the new location.
 a. Click the **Options** menu; then select **Copy Data Directory**.
 b. Select the new location. Create a new folder if necessary.
 c. Click **OK**.
 d. Click the **Options** menu; then select **Change Data Directory**.
 e. Select the directory to which the data was copied.
 f. Click **OK**.
3. Ensure that the files were moved correctly; then delete the data directory on the local computer if it is no longer needed.

Changing Cerberus' Data Directory Location (Other Computers)

The data files from additional computers can be moved from a computer to a network. Additional computers can access the data files by changing the Cerberus data directory for each PC.

➢ To change the data directory location (other computers)

1. **Start** Cerberus.
2. Export any projects that need to be added to any existing data directory.
3. Click the **Options** menu; then select **Change Data Directory**.
4. Enter/select the network directory to be used as the destination.
5. Import any projects that were previously exported.
6. Restart Cerberus.

Troubleshooting Installation

The following table provides information on troubleshooting Cerberus installation.

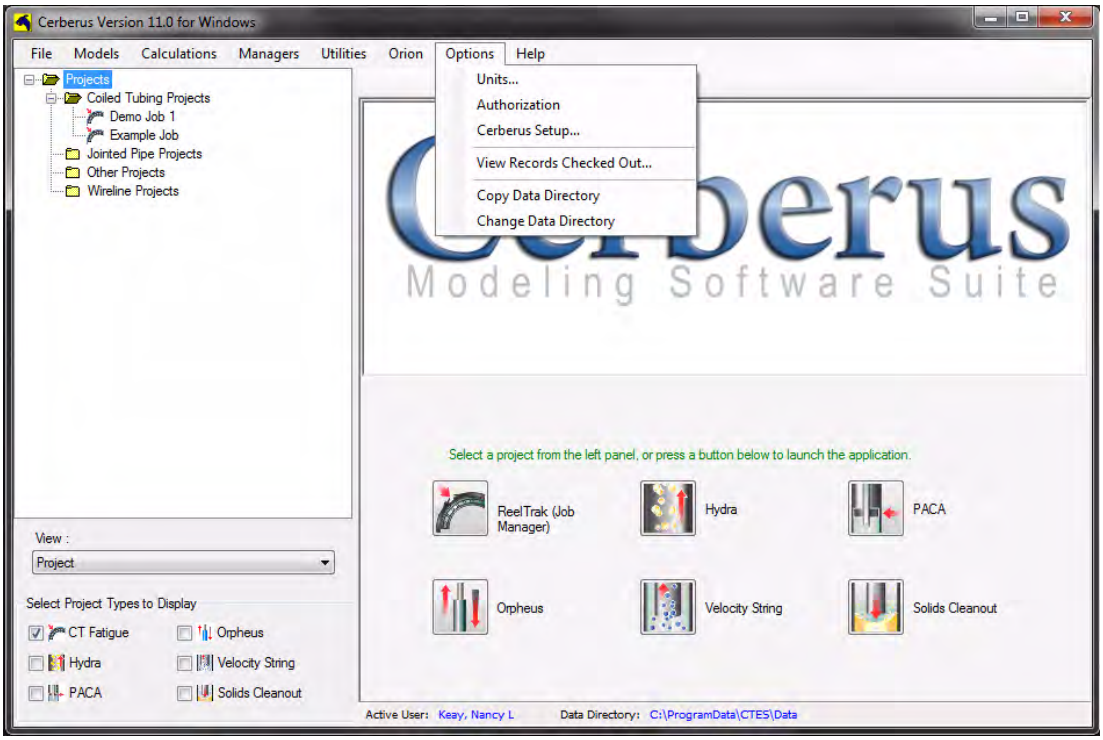
Situation	Possible Cause	Action
I receive an error message about not being able to copy a file.	Administrator rights for Windows may be needed, or the file may be in use.	Try logging out of Windows; log back in as a user with administrator rights. Then run the setup program again. Try closing all running applications, then run the setup program again.

Chapter 2
System Setup

Cerberus is configured with default values. However, key preferences can be changed with system administration rights.

Using the System Setup Options

Use the Cerberus **Options** menu-bar item to configure preferences.



Managing Units

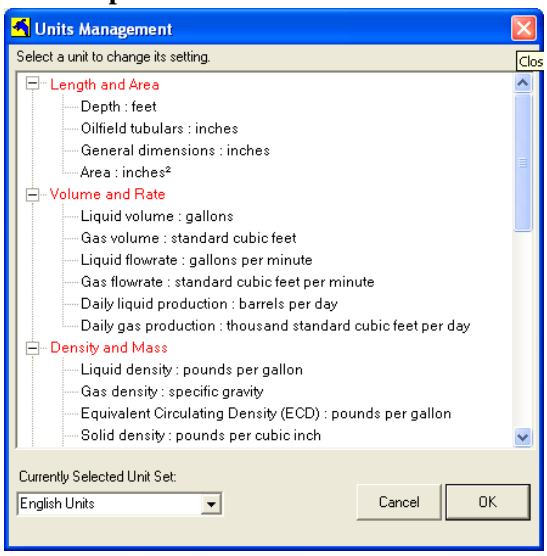
The Units Manager allows the user to determine whether units are specified as a unit “set,” or individually.

Unit Sets

When units are selected as a set, every unit displayed in the software conforms to the designated setting. In this “all or nothing” scenario, no unit may be individually specified.

➢ To specify a unit set

1. Select **Options** > **Units** on the menu bar.



2. Within the **Currently Selected Unit Set** field, select, from the dropdown, the unit set desired. The user may select from a list of pre-defined and/or user-defined unit sets. NOV CTES pre-defines three unit sets: English, Metric, and Canadian Metric.

To define unit sets, see the final, optional, step under Individual Units.

For example, if English is selected as the unit-set type, then each individual unit uses the pre-defined English-unit-set value.

NOV CTES-defined unit sets (English, Metric, or Canadian Metric) may not be deleted.

3. Click **OK** to accept the change or **Cancel** to reject the change.

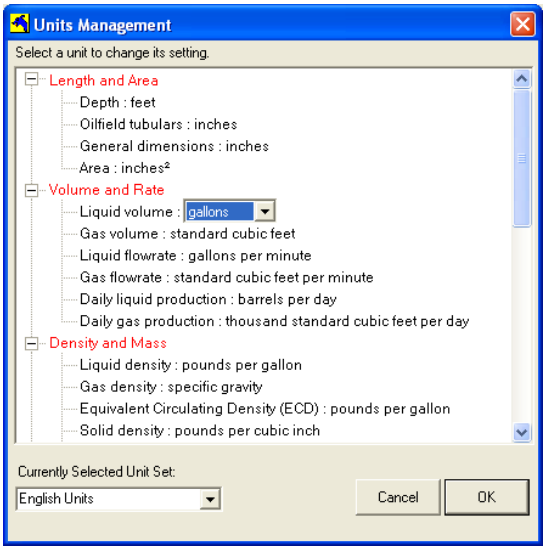
Individual Units

Units may be specified individually. Individual units may be saved and may also become part of a new unit set (though they may not become part of an existing unit set).

➢ To specify an individual unit

1. Select **Options** > **Units** on the menu bar.

2. Select the item for which unit-change is desired.



3. Use the associated dropdown to select the desired unit.

 (The **Currently Selected Unit Set** specification becomes “undefined.”)

4. Click **OK** to accept the change or **Cancel** to reject the change.

5. (Optional) Click **Save** to provide a new name for the unit set.

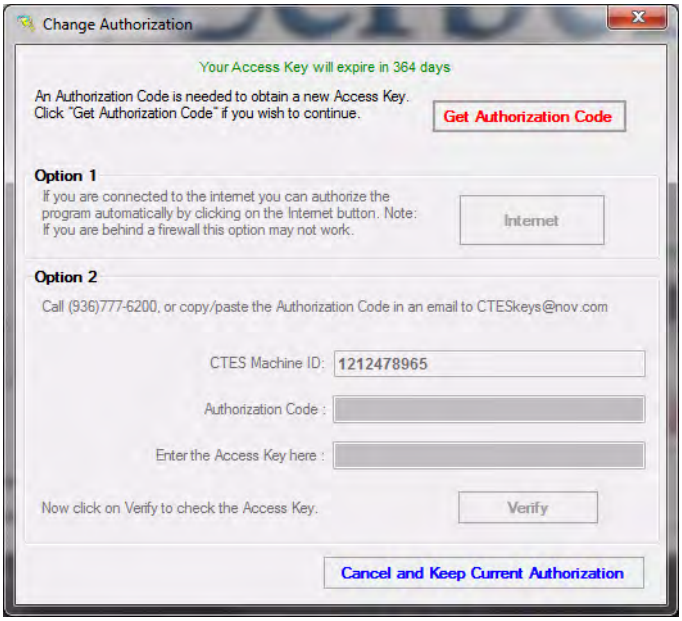
Changing Authorization

The authorization level determines the option availability in Cerberus. Typically, an authorization level is good for the term of the current Cerberus license period. To reflect new options purchased, or if the current authorization expiration is near, the authorization can be changed.

➢ To change authorization

1. Select **Options** > **Authorization...**.

2. Click **Get Authorization Code**.



3. If connected to the Internet, use Option 1 to obtain authorization over the Internet. (Note that if behind a firewall, this option may not work.)

 Click the **Internet** button, enter the contact information, and click **OK**. Text must be entered in all of the fields.

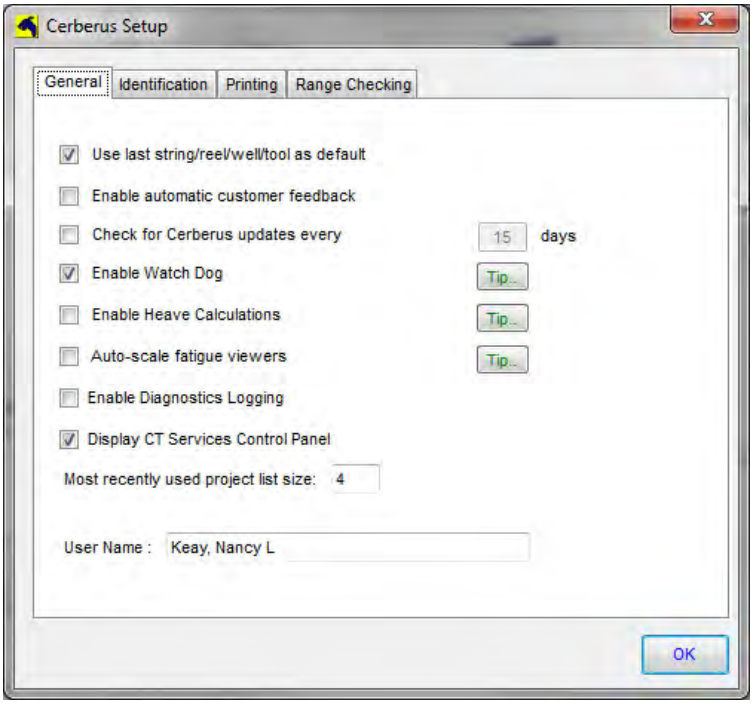
4. If not connected to the Internet, or if the Internet authorization fails, use Option 2 to obtain authorization. If possible, it is best to remain in front of the computer when contacting NOV CTES.

 Contact NOV CTES and supply the Authorization Code on the screen. Enter the corresponding Access Key provided and click **Verify**.

Using Cerberus Setup

➢ To Change Cerberus Setup

1. On the **Options** menu, select **Cerberus Setup...**.



The Cerberus Setup display screen contains four tabs.

2. Within the General tab, check/toggle the associated boxes to achieve the described effect:
 - **Use last string/reel/well/tool as default** – automatically selects the last item viewed by the user upon next startup.
 - **Enable automatic customer feedback** – activates customer-feedback management, allowing, after 182 days, a form to appear upon Cerberus startup.
 - Within the form, enter contact information and click **Yes** to send feedback information – in the form of a Cerberus usage datalog – to NOV CTES.

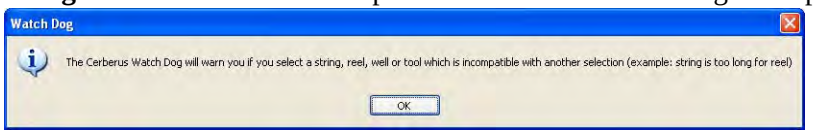
 OR

 - Within the form, click **No** to automatically deselect the checkbox. No information is sent to NOV CTES, unless the user re-checks the box.

 OR

 - Within the form, click **Later** to close the feedback information dialog. The form will again appear when the user re-opens Cerberus.
 - **Check for Cerberus updates every** – activates Cerberus' automatic check for software updates according to the number of days entered in the associated field.

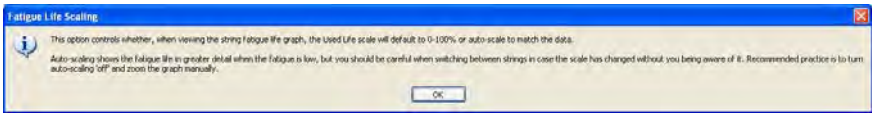
- **Enable Watch Dog** – Click the associated Tip button to view the following description:



- **Enable Heave Calculations** – Click the associated Tip button to view the following description:



- **Auto-scale fatigue viewers** – Click the associated Tip button to view the following description:



- **Enable Diagnostics Logging** – activates diagnostics logging.
- **Display CT Services Control Panel** – activates the viewing of the CT Services Control Panel, which should become visible on the right-hand side of the main CerberusNET window.
- The **User Name** is automatically populated. This may be changed.

3. Within the Identification tab, optionally enter user information.

4. Within the Printing tab, enter any information to be included on reports, such as disclaimers, logos, and so forth.

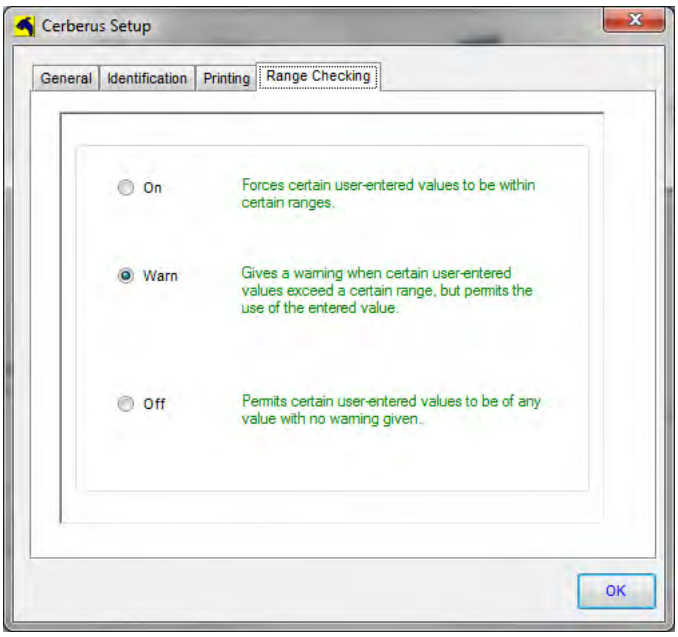
To add a logo, click the Browse button adjacent to the Logo field and navigate to the desired logo-image file.

5. Within the Range Checking tab, click (to toggle) the type of user-value-entry restriction desired:

 a. **On**: Forces certain user-entered values to be within certain ranges.

 b. **Warn**: Gives a warning when certain user-entered values exceed a certain range, but permits the use of the entered value.

 c. **Off**: Permits certain user-entered values to be of any value with no warning given.



Viewing Records Checked Out

When a user opens a well, tool string, project, or other item, Cerberus checks out that record. When the user is done with the item, Cerberus checks it back in.

If a Cerberus session ends unexpectedly, due to a power failure or other problem, Cerberus may not check in a record properly or a user may have a record checked out on a portable computer for an extended period of time. This keeps other users from viewing and editing the item. In such a situation, the user can undo a record check-out.

Undoing a Record Check-out

The following procedure outlines the steps for undoing a check-out on a record.



Confirm that a user is not actually using a record before undoing the check-out. Otherwise, the user will not be able to save any changes to that record and those changes will be lost.

➢ To Undo a Record Check-out

1. From the menu bar, select **Options** > **View Records Checked Out...**.

2. Click the record that is checked out.

3. Click **Undo Check Out**, then **Yes** to confirm.

Copying a Data Directory

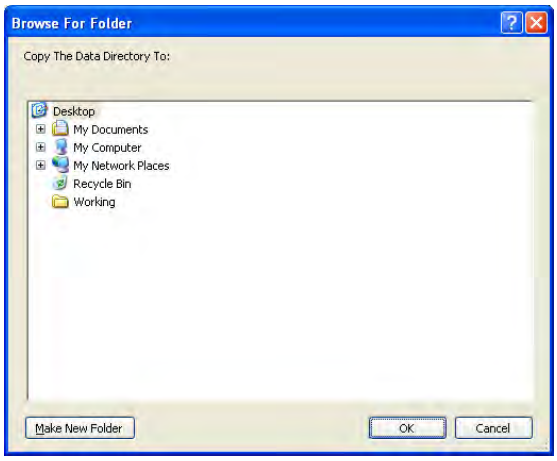
The data directory is where Cerberus stores all of its data files. The directory path and name are visible on the bottom of the main Cerberus menu.

Copying a data directory does not “zip” (or compress) the data like in a backup. It simply makes a copy of a data directory in a new location.

The target directory (the directory copied to) must be empty. If any file exists in the directory prior to the copy process, the copy attempt fails.

➢ To copy a data directory

1. Select **Options** > **Copy Data Directory** from the menu bar.

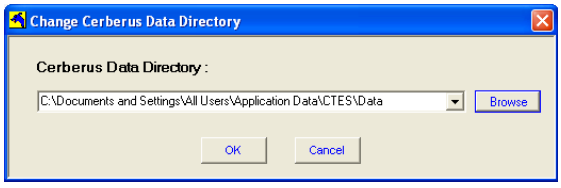


2. Use the Windows-like tools to select a location to which to copy the data directory.

Changing a Data Directory

➢ To change a data directory

1. Select **Options** > **Change Data Directory** from the menu bar.



2. Click **Browse** to navigate to the desired location.

3. Use the Windows-like tools to select a Cerberus data directory.

Chapter 3
Using Cerberus

This section describes basic Cerberus-software purpose, access, and navigation.

Getting Started

Cerberus is primarily a tool-deployment planning system, providing modeling of:

- cumulative mechanical and hydraulic forces involved in well intervention.
- pressure and flowrate calculations for workover operations.

Cerberus helps determine:

- if target depth is reachable, and
- the most feasible conveyance method and equipment configuration to complete the job and to return the tools safely and efficiently to surface.

This section explains basic concepts related to starting and using Cerberus.

Starting Cerberus

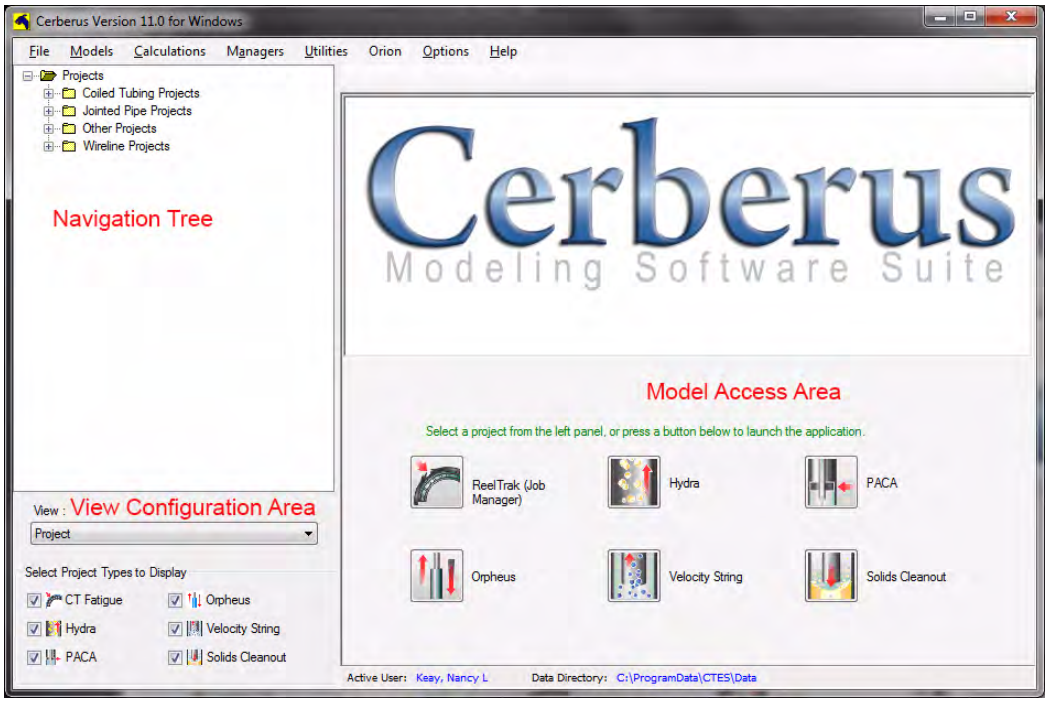
➢ To start Cerberus

1. Click **Start** in the task bar.
2. Click **Programs** > **CTES** > **Cerberus 11.0**, or click the **Cerberus** icon on the desktop.
3. If this is the first time Cerberus is run, the user must obtain authorization for this copy of Cerberus. See Starting Cerberus and Obtaining Authorization for more information.
4. When the application is authorized, the Cerberus Main Menu appears.

The Main Menu (Home Page)

The Cerberus Main Menu screen appears after the initial Cerberus startup.

Cerberus Main Menu



This interface is the main “hub” through which the user works with Cerberus.

Within this main hub, Cerberus focuses primarily on the following:

- Projects: these provide a framework for organizing the various elements that drive the calculation process, which ultimately provides the user with decision-making tools in the job-design process.
- Managers/Editors: the data-gathering, holding and administration areas for the various contributors to the calculation process.

- Models: the applications from which the various components that are used in projects – including the conveyance methods (coiled tubing strings, jointed pipe, and/or wireline) and specifications used to determine feasibility of satisfactory job completion – are “driven” to the calculated outcome.

The following table describes the Cerberus models and the conveyance methods associated with them.

Cerberus Model	Conveyance Method
Reel-Trak (Job Manager)	Coiled Tubing
Orpheus (forces)	Wireline, Slickline, Coiled Tubing, Jointed Pipe
Hydra (hydraulics)	Coiled Tubing, Jointed Pipe
Velocity String	Coiled Tubing, Jointed Pipe
PACA (packer and completion analysis)	Coiled Tubing, Jointed Pipe, No Pipe
Solids Cleanout	Coiled Tubing

The Main-menu Layout

Cerberus’ main menu, or “home” page, is divided into sections, as follows:

Navigation Tree

The navigation tree, found within the upper left-hand portion of the display screen, provides a Windows-like hierarchy of folders and files, from which the user can navigate through and access (depending on Cerberus licensing) projects, wells, coiled tubing strings, wirelines, or jointed pipes.

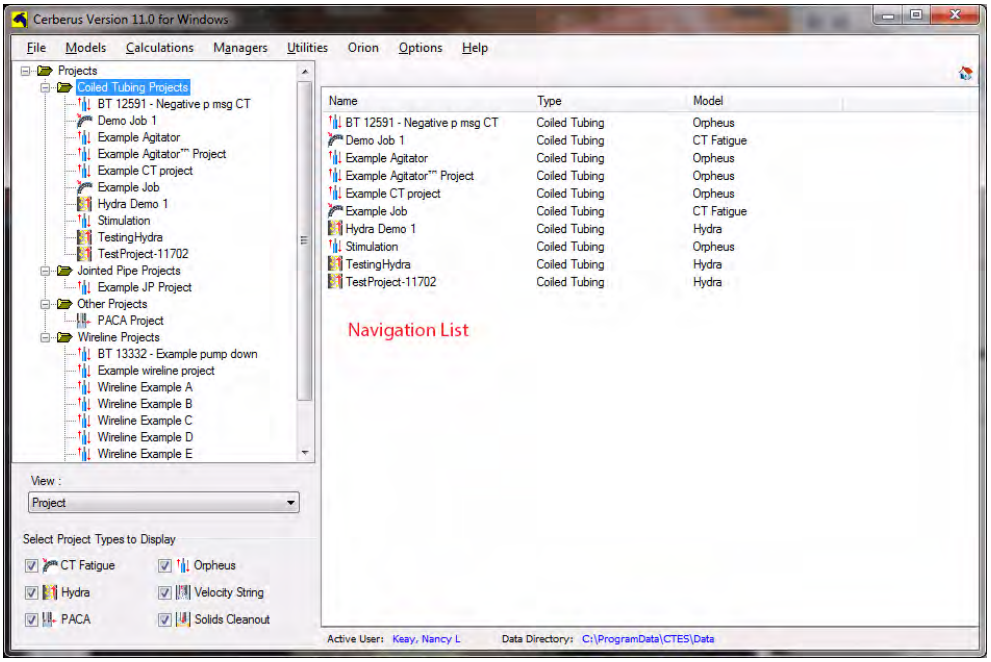


Folders are internal to Cerberus and unrelated to folders in Windows Explorer®.

Folder availability is dependent on the user’s selection(s) in the View Configuration Area, and on the Cerberus licensing for the user’s site.

Navigation List

When the user selects a folder within the navigation tree, a navigation list opens on the right-hand side of the display screen.



File Management

The user familiar with Microsoft Windows operation will find that the navigation list, when used in conjunction with the navigation tree, is very similar to the Windows Explorer file management system.

Click (to select) a single item, or use the *Shift* or *Ctrl* key to select multiple projects and folders in the navigation list. The user may “drag and drop” items (left-click and drag; release to drop) as follows:

List to list: multiple items may be dragged and dropped into a folder within the navigation list.

List to tree: multiple items may be dragged and dropped from the navigation list to the navigation tree.

Tree to tree: a single project may be dragged and dropped within the navigation tree.

Tree to list: a single project may be dragged and dropped from the navigation tree to a folder in the navigation list.

For user-defined folders only, right-click the folder to add, re-name, or delete it. System-defined folders cannot be renamed or deleted.

View Configuration Area

In the lower left-hand portion of Cerberus’ main menu lies the View Configuration Area. The area is divided into two sections: **View** and **Select Project Types to Display**.

Select among **View** types (available depending on Cerberus licensing) by clicking the desired view from the associated dropdown.

Determine the availability of navigation-tree **Project** information by toggling/clicking selections in the **Select Project Types to Display** area.

Project-type availability is dependent upon the Cerberus licensing available to the user.

Model Access Area

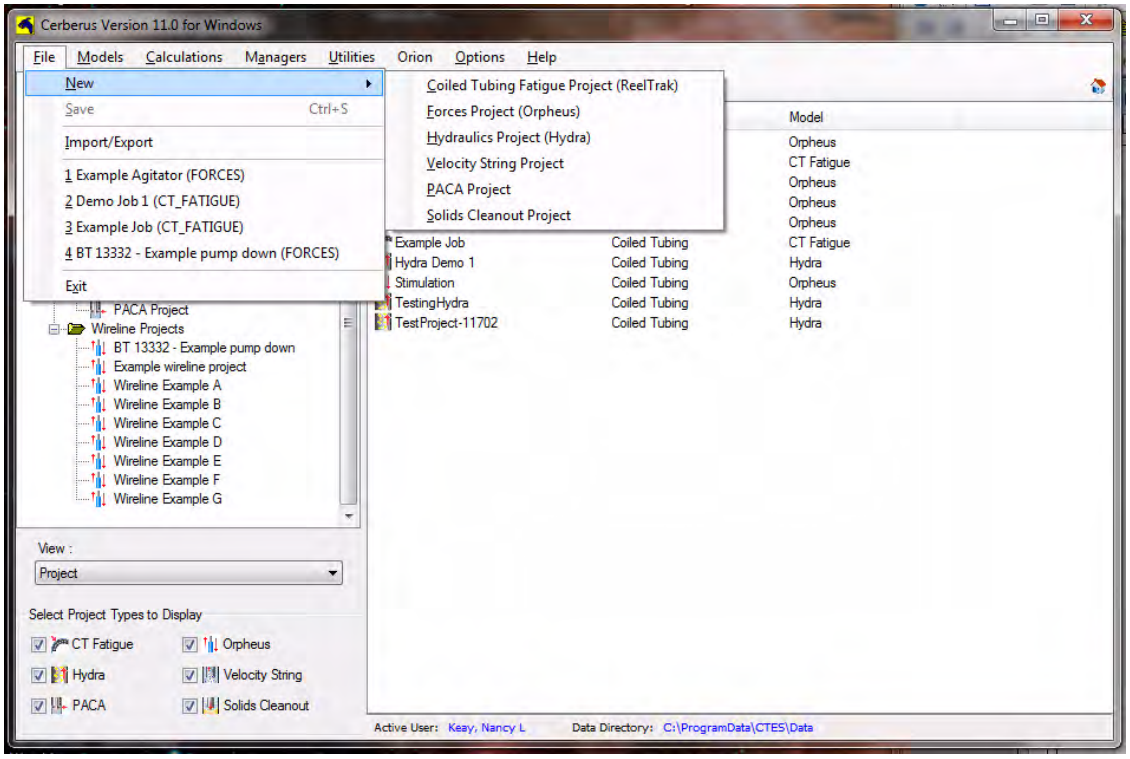
Launch a Cerberus modeling application by clicking one of the available (depending on Cerberus licensing) model buttons.

Menu Bar

Cerberus uses a typical Windows-style menu bar for navigation.

1. The **File** menu allows the creation of a new project, saving of a project, "undoing" a change, importing/exporting of a project, and exiting from the Cerberus application.

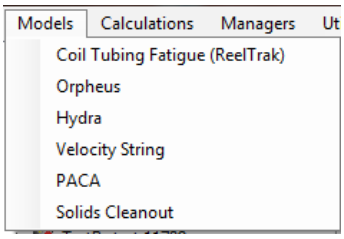
 Cerberus File Menu



Some File menu-bar items are available only until the user accesses a project, well, coiled tubing string, wireline, or jointed pipe.

2. The **Models** menu-bar item allows selection from among the Cerberus models that are also available from the Model Access Area.

Cerberus Models Menu



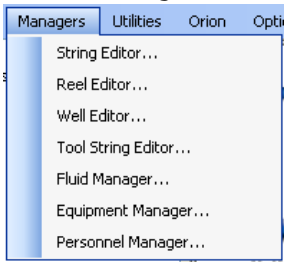
3. The **Calculations** menu-bar item provides access to **Achilles** (coiled tubing fatigue calculations) and to **Hercules** (coiled tubing stresses calculations (including bust, collapse, tension, and compression limits)).

Cerberus Calculations Menu



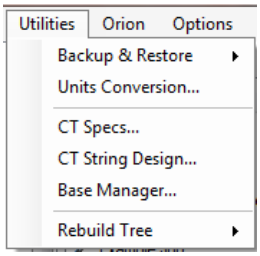
4. The **Managers** menu-bar item provides access to the **String Editor** (CT), **Reel Editor** (CT), **Well Editor**, **Tool String Editor**, **Fluid Manager**, **Equipment Manager**, and **Personnel Manager**.

Cerberus Managers Menu



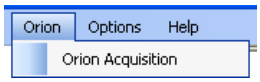
5. The **Utilities** menu-bar item allows the user to **Backup/Restore** a data directory. It also provides access to **Units Conversion**, **CT Specifications**, **CT String Design**, **Base Manager** (Cerberus-module data analysis in relation to jobs), and the ability to rebuild the **Project Tree** or the **String Tree**.

Cerberus Utilities Menu



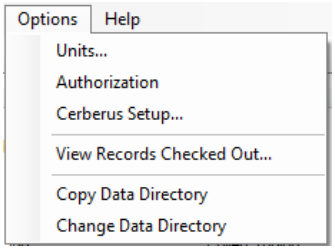
6. The **Orion** menu-bar item allows the user to open OrionNET software (if available) and begin data acquisition.

Cerberus Orion Menu



7. The **Options** menu-bar item allows the user to change **Unit** settings; change software **Authorization**, as in a Cerberus software-usage upgrade; edit the **Cerberus** software **Setup**, as in altering default settings, inserting company identification information in printed documents and setting value ranges; **View** database **Records** currently **Checked Out**; and **Copy** or **Change** a **Data Directory**.

Cerberus Options Menu



When a Cerberus screen layout or feature does not appear as the user expects it to, it is best to check status of the various options available in **Cerberus Setup**.

Many important Cerberus settings are found in **Cerberus Setup** that can affect the behavior and appearance of the software. The current defaults are checked upon initial access, and Cerberus retains new settings.

a. Cerberus Setup > General tab

The General tab-settings include those listed below. Click (to check/activate or to uncheck/deactivate on alternate clicks):

1. **Use last string/reel/well/tool as default**

2. **Enable automatic customer feedback**

3. **Check for Cerberus updates every [n] days**. Input the number in the associated field.

4. **Enable Watch Dog**. The Cerberus Watch Dog displays a warning if a string, reel, well, or tool is selected that is incompatible with another existing selection. Example: the string is too long for the reel. (Click **Tip** to see a message similar to this one.)

5. **Enable Heave Calculations**. Heave calculations estimate the additional fatigue due to coiled tubing movement at the reel and gooseneck caused by a wave compensation system. Checking the box causes an additional tab to be displayed on the Job Log screen of Reel-Trak. Heave is configured on a job-by-job-basis.

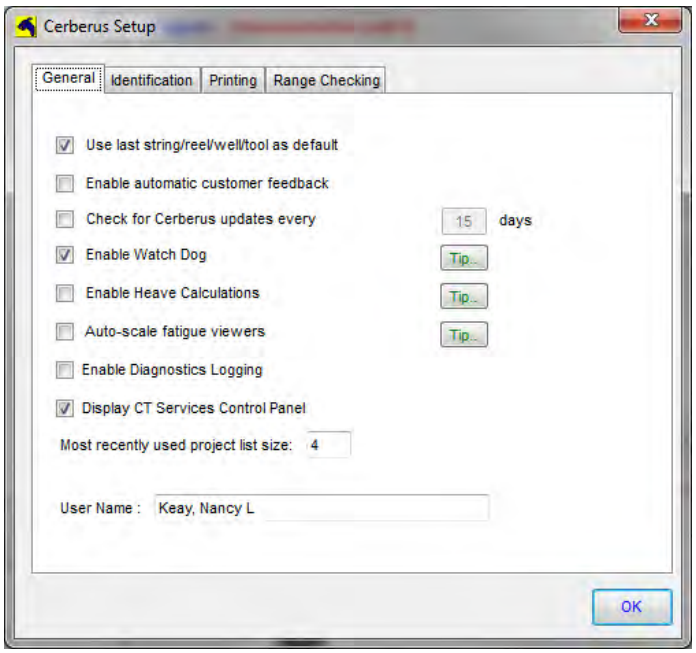
6. **Auto-scale fatigue viewers**. This option controls whether, when viewing the string fatigue life graph, the Used Life scale defaults to 0-100% or auto-scales to match the data.

Auto-scaling shows the fatigue life in greater detail when the fatigue is low. Care should be taken when switching between strings in case the scale has change without user awareness. The recommended practice is to turn auto-scaling “off” and to zoom the graph manually.

7. **Enable Diagnostics Logging**

8. **Display CT Services Control Panel**. When this option is checked, the CT Services Control Panel appears on the Cerberus main screen when the user selects a job from the Navigation Tree, or in Reel-Trak on the Job Setup and/or Post Job Analysis tab.

Cerberus Options Menu > Cerberus Setup > General Tab



b. Cerberus Setup > Identification tab

Use the Identification tab to fill in optional information used as input to order forms and other similar uses where the company identity is required.

c. Cerberus Setup > Printing tab

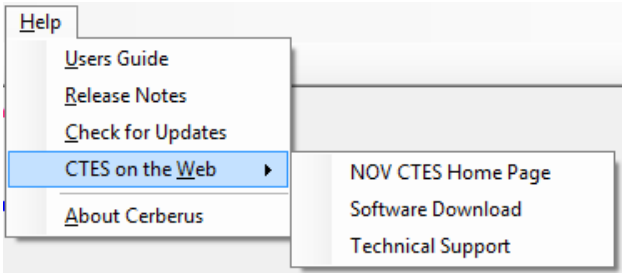
Use the Printing tab to set up the parameters that will appear on reports.

d. Cerberus Setup > Range Checking tab

Use the Range Checking tab to choose among types of range warnings for user-entered values.

8. The **Help** menu-bar item provides access to the Cerberus **User Guide** or **Release Notes**; allows the user to **Check for** software **Updates**, provides access to NOV **CTES on the Web** to view the home page and/or for **Software Download**; and provides information **About** the **Cerberus** version number and the identification of the software user.

Cerberus Help Menu



Shortcut Keys

The following shortcuts are useful for navigating in Cerberus:

Ctrl+E = Expand the entire navigation tree.

Ctrl+Q = Collapse the entire navigation tree.

Ctrl+R = Restore the Cerberus display screen to its default size and location.

Ctrl+S = Save the changes made.

Ctrl+Z = Undo the changes made to a well, string, reel, or tool string for a project.

When Cerberus is closed and re-accessed, the last View (project, well, CT string, jointed pipe, or wireline) used (see "View Configuration Area" on page 23) appears. The upper right-hand area of the display screen shows related project information, and the lower right-hand area of the display screen shows the related model.

Chapter 4
Reel-Trak

This section describes the process involved in analyzing fatigue in coiled tubing jobs. This analysis may occur pre-job, post-job, or in real time (during actual operations at the wellhead). In the case of post-job analysis, the model's calculation tools calculate actual fatigue.

About Coiled Tubing (CT) Fatigue Tracking with Reel-Trak

Cerberus' Reel-Trak interface is organized to support a structured approach to fatigue tracking. This approach utilizes four Reel-Trak "modes," which describe the various states in which a job can exist during the process. These include Creating a New Job, Job Setup, Fatigue Tracking, and Post Job Analysis. This mode-based methodology is beneficial for real-time operations, as well as for tracking fatigue after job completion.

Before accessing and using Reel-Trak functionality, please select, from the main Cerberus menu-bar, ***Options*** *>* ***Cerberus Setup*** *to specify Cerberus setup options that can affect the user's experience with Reel-Trak.*

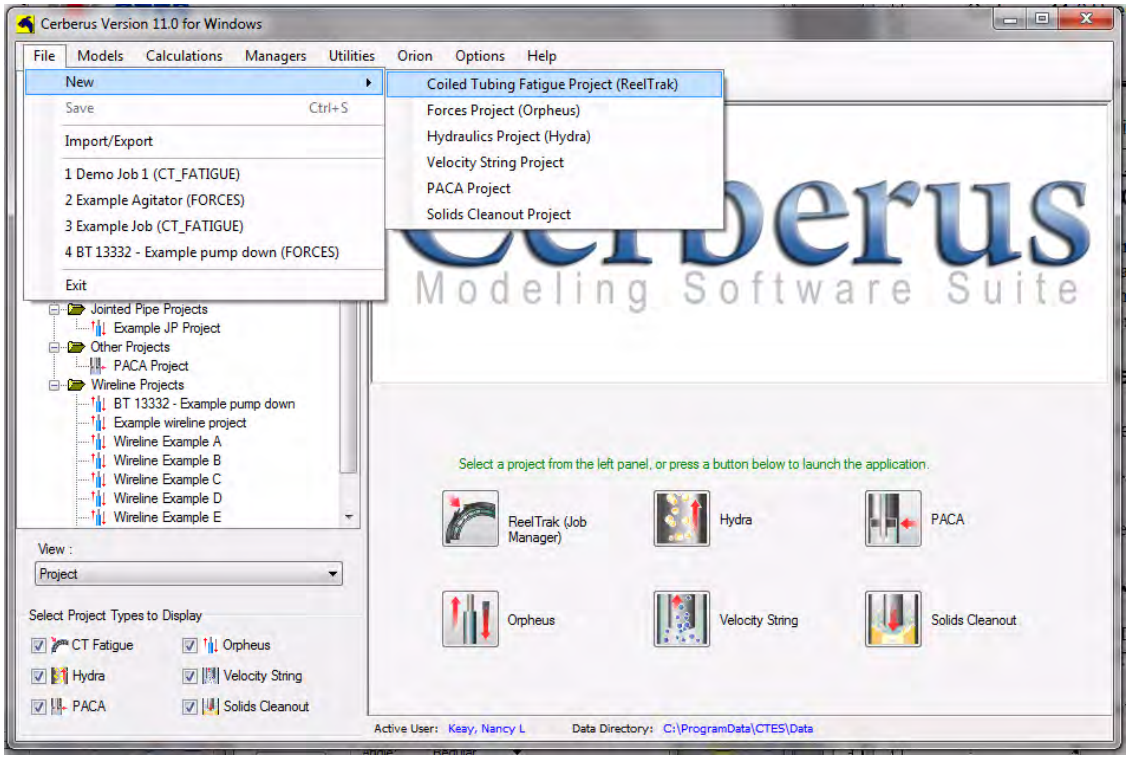
Creating a New Job

There are several ways to create a new job.

➢ To create a new job

Choose from one of the following approaches:

1. From the main Cerberus menu, select **File > New > Coiled Tubing Fatigue Project (ReelTrak)**.



a. The Job Wizard is launched at this point **IF**:

- The new job is requested immediately after Cerberus or Reel-Trak startup.

 OR

- **Show Wizard on start** is toggled within the Job Wizard.

b. Otherwise, the New Job dialog appears.

2. From the Reel-Trak menu bar, select **File > New Job**.

 This launches the Job Wizard or the New Job dialog, with behavior identical to the first option, above.

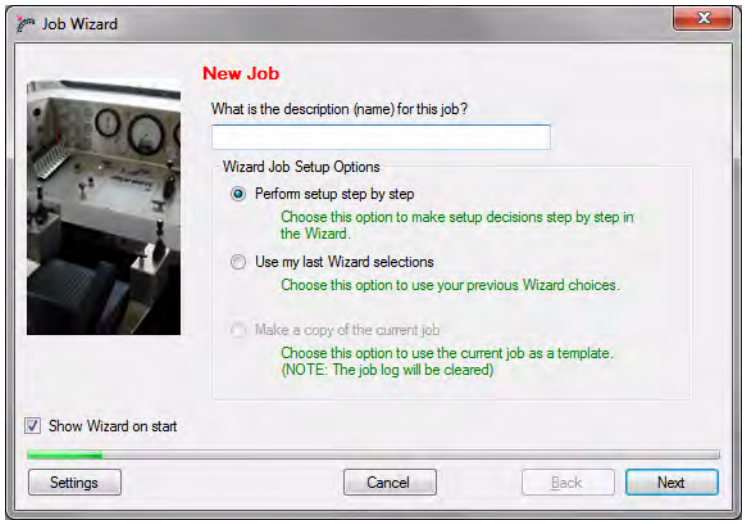
3. From the Reel-Trak menu bar, select **Help > Job Wizard**.

4. From the Reel-Trak toolbar, select the **Job Wizard** icon  .

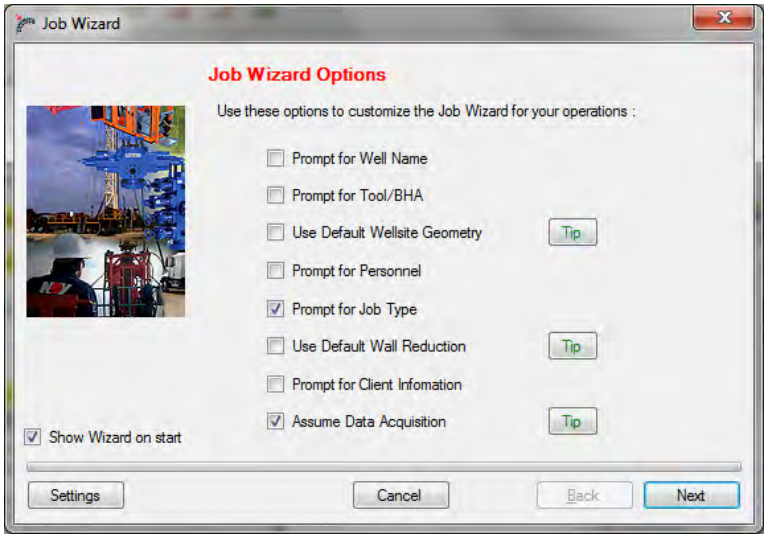
Using the Job Wizard

➢ To use the Job Wizard

1. Access the Job Wizard (see "To create a new job" on page 30).



2. (Optional) Customize **Job Wizard Options** by selecting the **Settings** button.



 Use the available **Tip** buttons for information about the related checkboxes; click **Next**.

3. Input a job description (name). (In each case here and below, select **Next** to advance through the wizard.)

4. Click to select/toggle **Wizard Job Setup Options.**

 Choose from among three options for creating a job.

 a. **Perform setup step by step**.

 1. Select a string. If the string is not available in the list, it must be uploaded to the computer from the appropriate source (coordinator, dispatch, engineering, or other). In the case of field operations where the string is not available, a string file may be created using the String Editor (select **Managers > String Editor** from the main menu bar).

 2. Select a reel. If the reel is not available in the list, upload a reel acquired from the appropriate source (coordinator, dispatch, engineering, or other) to the computer. In the case of field operations where a reel file is not available, it may be created using the Reel Editor (select **Managers > Reel Editor** from the Cerberus main menu bar).

 3. Input/select wellsite geometry: **Gooseneck Length**; **Gooseneck Radius**; **Reel to Gooseneck**; **Top of Injector to Zero Depth**; **Tool Length**; **Depth measured from end of tool/coil.**

 4. Select a job type. If the selected job type potentially affects the wall thickness of the pipe (such as an acid or abrasive job type), a default **Wall Reduction** appears in the related field. If no acidic or abrasive effect is typical with the job type selected, no default **Wall Reduction** appears.

 To set up the default **Wall Reduction**, edit the **Job Type**. See “Setting Up a CT Job” on page 34.

 To turn on/off the wall-reduction application, select, from the Reel-Trak menu bar, **Options > Preferences**. Then select the Fatigue Tracking tab, if it is not already selected, and use the available parameters.



The Job Wizard behavior can vary dramatically based on the Settings selected.

 b. Click to select/toggle **Use my last Wizard selections**.

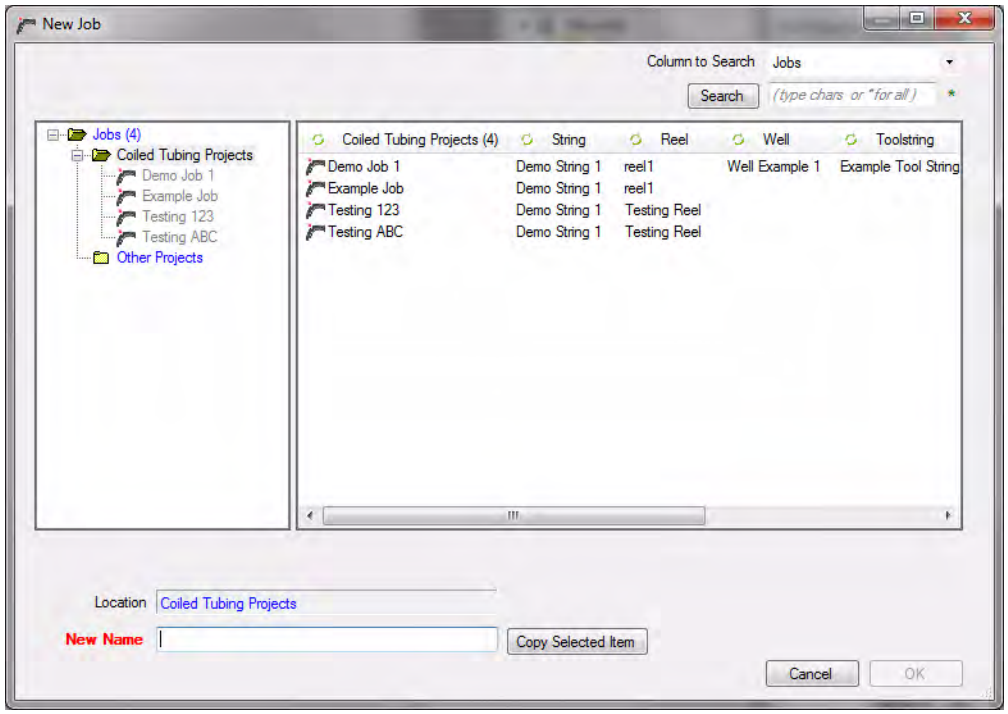
 c. Click to select/toggle **Make copy of the current job**. This option uses the current job as a template, and the job log is cleared.



Previous selections are stored and can be used to create a quick job.

Using the New Job Dialog

In the scenario described here, the New Job dialog is accessed as described in "To create a new job" on page 30.



➢ To create a new job using the New Job dialog

1. Use the left-hand panel to select the folder in which to save the job; otherwise the job is saved in the Jobs folder.

 The user's selection determines the folder in which the new job is located, as indicated in the un-editable **Location** field.

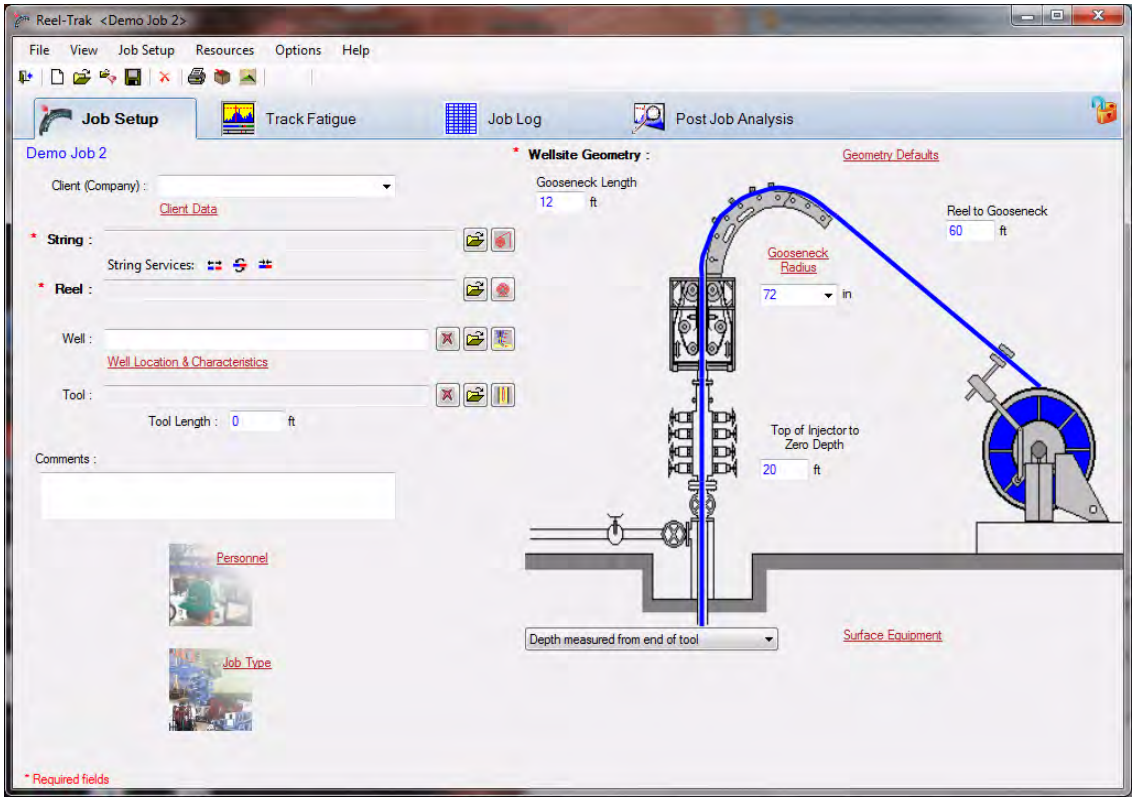
 The existing jobs within the selected folder are displayed both in the left-hand panel and in the right-hand panel.

2. (Optional) Select a job from either the left-hand or right-hand panel and click **Copy Selected Item** to copy the job name.

3. Input a name for the new job in the **New Name** field.

4. Click **OK**.

The job is loaded/displayed in Reel-Trak with the Job Setup tab open and ready for further input.



Setting Up a CT Job

Reel-Trak's Job Setup tab is the launchpad for a Cerberus job where job components and parameters such as the CT String and Reel are defined or edited. This panel/tab is used to configure a Reel-Trak job for fatigue tracking, either in real-time or post-job. Wellsite geometry, as well as job type and job personnel, are configurable from this main panel.

➢ To set up a CT job

1. Open the CT job (using one of the following methods):

 a. From the Cerberus main menu, double-click an existing CT job from the left-hand panel.

 b. From the Cerberus main menu, select the **ReelTrak (Job Manager)** button. .

 1. Double-click a CT job from the Select a Job dialog.

 2. Select the Job Setup tab.

 c. From Reel-Trak's menu bar, select **File > Open Job**.

 d. To view existing jobs that use a particular string (and to open one of these), select the **Open using String Filter** toolbar item .

From the resulting dialog, click the dropdown to view available strings and to select a CT string.

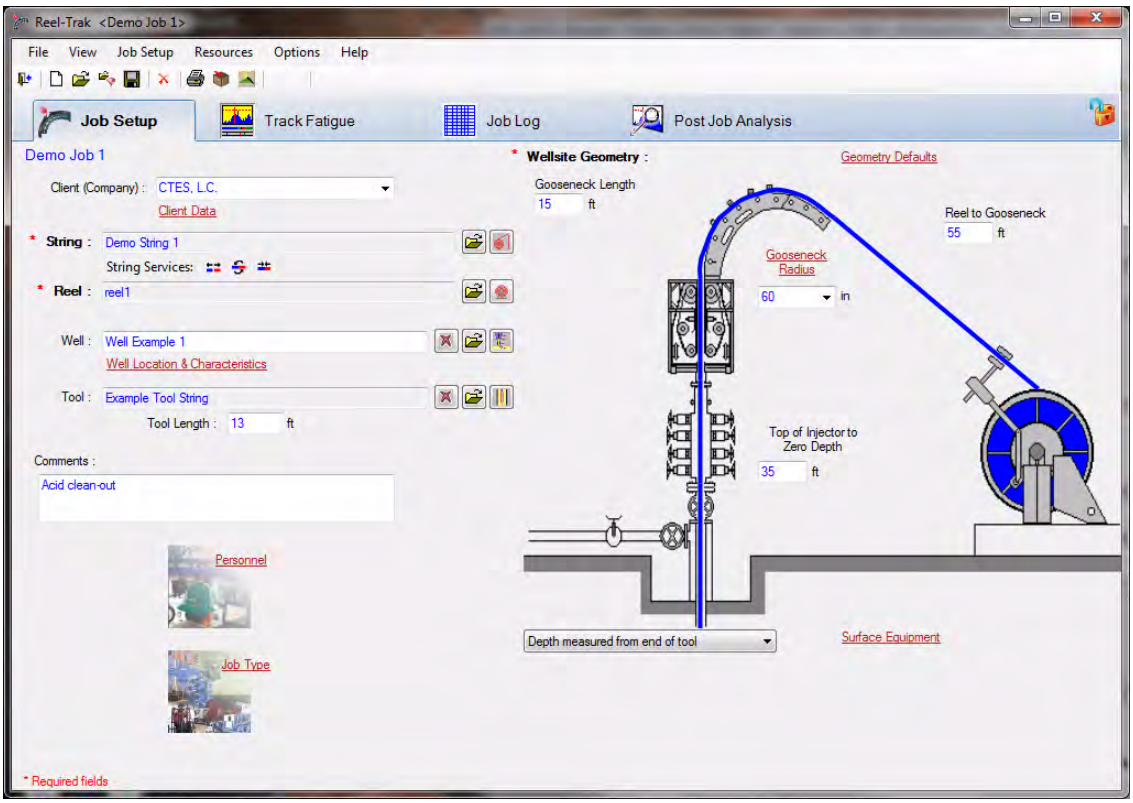
2. If the Post Job Analysis tab is displayed when the job is opened, the job has already been completed, and fatigue has been applied.

 a. The user may review the fatigue applied during the job, and analysis of the health and nature of fatigue life used, and graphs of the data used to calculate fatigue.

 OR

 The user may "undo" the running of fatigue for a string via the String Editor (see "Printing String Data" on page 234). In this scenario, the tracked fatigue for the job is removed from the string.

3. If not already selected, select the Job Setup tab in Reel-Trak.



4. If the job is locked, note the following before proceeding:

 If a job is complete, the job may not be unlocked. The only way to unlock a complete job is to undo the job through String Editor. See "Printing String Data" on page 234 for additional information.

The job must be unlocked in order to edit/set up parameters. The job may, however, be reviewed in the Track Fatigue mode even if it is locked.

➢ To unlock the job if it is not complete

- Click the "lock" icon in the upper right-hand corner of the interface to unlock the job.

OR

- Select **File** > **Locked/Archived Status** to use the Locked/Archived Status dialog to unlock the job.



5. Input data related to the job. Fields with associated red asterisks are required; others are optional. Required fields are needed for fatigue to be successfully tracked.

Not selecting a **String** *or* **Reel** *does not prevent the job from being saved.*

Personnel *must have a role selected in order to be saved.*

a. **Client (Company)**:

- Use the dropdown to select an existing client/company.
- To input and save client/company information on a related data-input form, click the **Client Data** hyperlink.

OR

Select **Job Setup** > **Client Data** from the menu bar.

b. ***String**:

- To select an existing string or create a new string via the Select a String dialog, click the Folder icon associated with the **String** field.

OR

Select **Job Setup** > **String** from the menu bar.

- Click the String Editor icon to access the String Editor (see “Using the String Editor” on page 215).
- To cut, reverse, or splice a string prior to the initiation of fatigue tracking, select one of the

String Services. When fatigue tracking has begun and the job is locked, these tools become disabled. (see "String Services" on page 254 for more information).

If the String field is empty, clicking the String Editor icon accesses the Select a String dialog.

c. ***Reel:**

- To select an existing reel or to create a new reel via the Select a Reel dialog, click the Folder icon associated with the **Reel** field.

 OR

 Select **Job Setup > Reel** from the menu bar.

- Click the Reel Editor icon to access the Reel Editor (see "Using Reel Editor" on page 327).

If the Reel field is empty, clicking the Reel Editor icon accesses the Select a Reel dialog.

d. **Well**:

- To remove an existing well from the job, click the Delete icon.

- To select an existing well or create a new well via the Select a Well dialog, click the Folder icon associated with the **Well** field.

 OR

 Select **Job Setup > Well** from the menu bar.

- To access the Well Editor, click the Well Editor icon (see "Using the Well Editor" on page 301).

The user also has the option of manually typing a well name. This allows users who do not need all the detail of a Cerberus well to easily enter important details about the well for the job.

- To input and save well information on a related data-input form, click the **Well Location & Characteristics** hyperlink.

 OR

 Select **Job Setup > Well Location & Characteristics** from the menu bar.

e. **Tool:**

- To remove an existing tool from the job, click the Delete icon.
- To select an existing well or create a new well via the Select a Toolstring dialog, click the Folder icon.

 OR

Select J**ob Setup > Tool** from the menu bar.

- To access the Tool String Editor, click the Tool String Editor icon (see "Using the Tool String Editor" on page 269).

f. **Comments**: Add comments as desired.

g. **Personnel**: To reveal the Personnel-related area, click **Personnel** within the Job Setup tab.

OR

Select **View > Personnel** from the Reel-Trak menu bar.

*Selecting **View > Personnel** also closes an open Personnel-related area.*



Within this area, use the dropdown(s) to select existing **Personnel** and existing **Roles** for the **Personnel.** Use the **Remove Person** icon to delete a listed person.

- To input and save personnel information on a related data-input form, click the **Personnel Manager** hyperlink.

 OR

 Select **Resources > Personnel** from the menu bar.

 See "Using Personnel Manager" on page 355 for additional information regarding using the Personnel Manager.

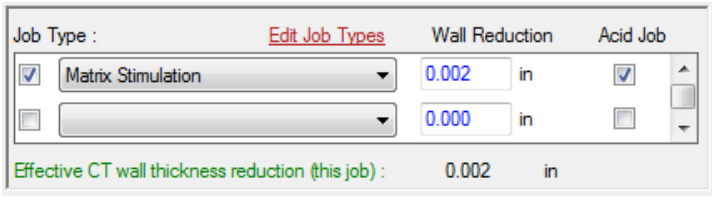
h. **Client Data**: To add a new client, edit an existing client's information, or delete a client, select **Job Setup > Client Data** from the menu bar.

i. **Job Type**: To reveal the Job Type-related area, click **Job Type** within the Job Setup tab.

OR

Select **View > Job Type** from the Reel-Trak menu bar.

*Selecting **View > Job Type** also closes an open Job Type-related area.*



- Within this area, select from existing **Job Types** using the associated dropdown(s). Selecting a

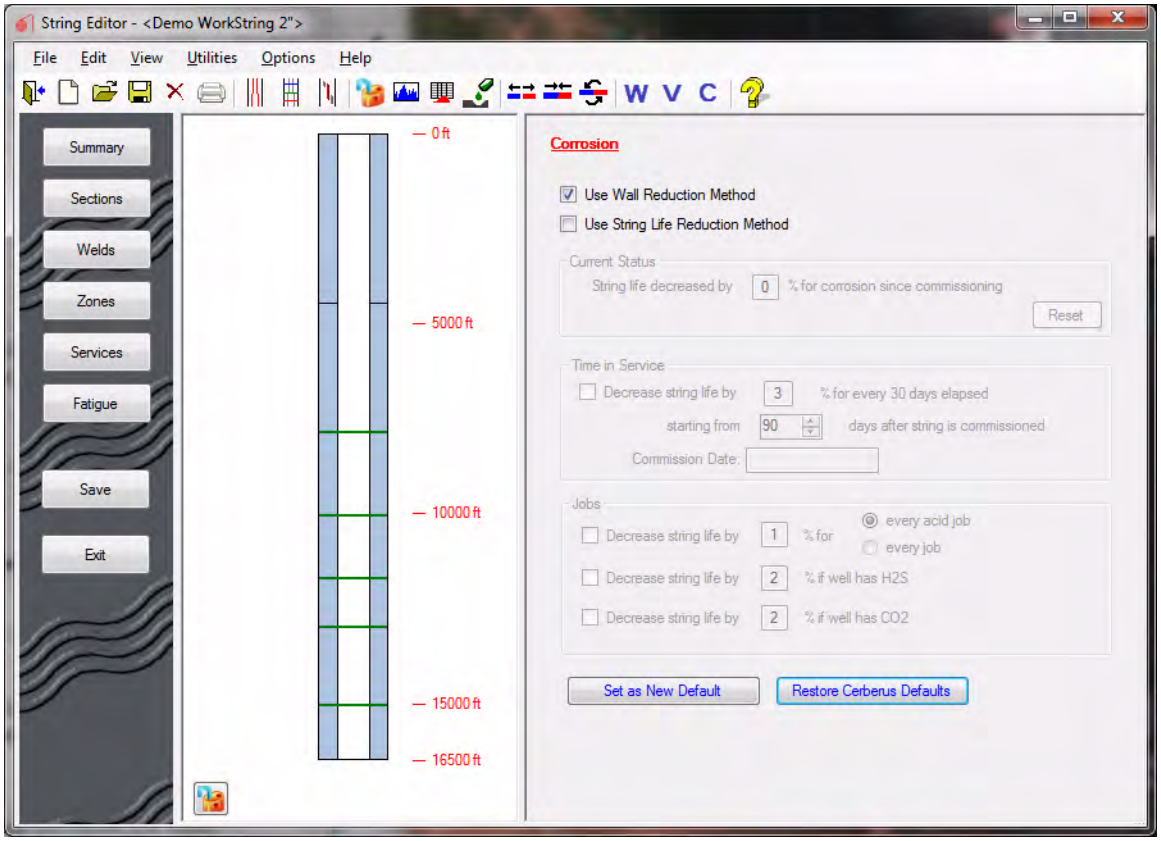
job type automatically checks the box in front of the **Job Type** field, fills in a default **Wall Reduction** value, and checks the box **Acid Job** if the job is acid.

- To input and save job-type information on a Default Job Types dialog, click the **Edit Job Types** hyperlink. Here, the user may add job types and define their **Wall Reduction** and **Acid Job** defaults. The user may also delete job types here.

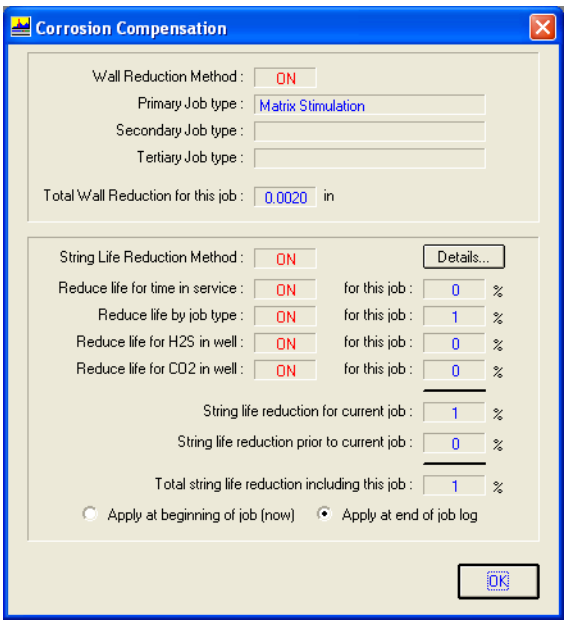
*The **Wall Reduction** value automatically populates in the **Job Type** area from the information entered via the **Edit Job Types** hyperlink (which accesses the Default Job Types dialog), but may be changed manually in the **Job Type** area.*

- The **Effective CT wall thickness reduction (this job)** is displayed immediately below these fields in the Job Setup tab.

The user can set the method of Corrosion calculation (Wall Reduction or String Life Reduction) through the String Editor. See "Entering Corrosion Settings for a String" on page 250 for additional information.



In Reel-Trak, a dialog containing the String Life Reduction and Wall Reduction statuses (on/off), life-reduction percentages, and wall reduction amount is available to the user. From the menu bar, select **Job Setup > Corrosion Compensation/Wall Reduction**.



j. ***Wellsite Geometry**: The current default parameters for the **Wellsite Geometry** are displayed.

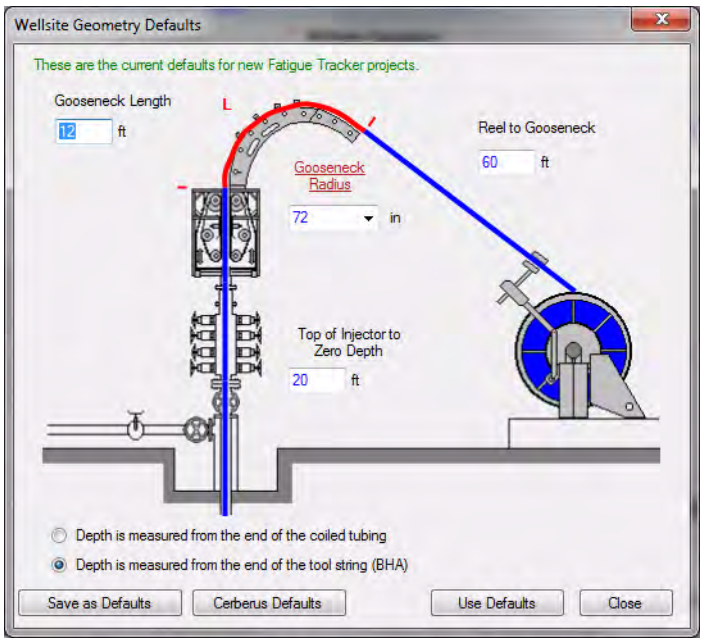
The Wellsite Geometry portion of this tab/panel becomes disabled when the job is locked. The assumption in operation is that the job has been set up and fatigue tracking is either in progress or will be in progress shortly.

- To view the **Injector** field within this area, select **View > Injector (Wellsite Geometry)**.

 Use the dropdown to select an injector, if desired.

 To edit injector properties, select **Resources > Equipment Manager**. See "Using Equipment Manager" on page 349 for additional information.

- Click **Geometry Defaults** to access the Wellsite Geometry Defaults dialog for the purpose of altering the defaults for new Fatigue Tracker projects.



Current defaults are displayed in the Wellsite Geometry Defaults dialog, including **Gooseneck Length**, **Gooseneck Radius**, **Reel to Gooseneck** measurement, and **Top of Injector to Zero Depth** measurement.

- To change the default settings for new Fatigue Tracker projects, change any or all of these numbers and select **Save as Defaults**. Click the **Gooseneck Radius** hyperlink to access the Gooseneck Arch Radius dialog, from which the user may **Add** a **New Radius**, **Delete** an existing radius, or select an **Existing** radius **value**. Click **Done** when tasks are complete.

 OR

 To fill the fields with Cerberus defaults, select **Cerberus Defaults**.

 OR

 To fill the fields with the user's previously saved defaults for new Fatigue Tracker projects, select **Use Defaults**.

- Click (to select/toggle) **Depth is measured from the end of the coiled tubing** or **Depth is measured from the end of the tool String (BHA)**.

- Select **Close** to exit the Wellsite Geometry Defaults dialog and return to the Job Setup tab of Reel-Trak.

- Alter any of the available number values in Wellsite Geometry: **Gooseneck Length**, **Gooseneck Radius**, **Reel to Gooseneck** measurement, and **Top of Injector to Zero Depth** measurement by typing directly into the field, or by selecting from available options when dropdowns are available.

k. **Surface Equipment**: To select the equipment package used on the job, click the **Surface Equipment** hyperlink.

OR

Select **Job Setup** > **Surface Equipment** from the menu bar.

If additional equipment must be defined, click the Equipment Manager button to access the Equipment Manager dialog. See "Using Equipment Manager" on page 349 for additional information.

Tracking Fatigue

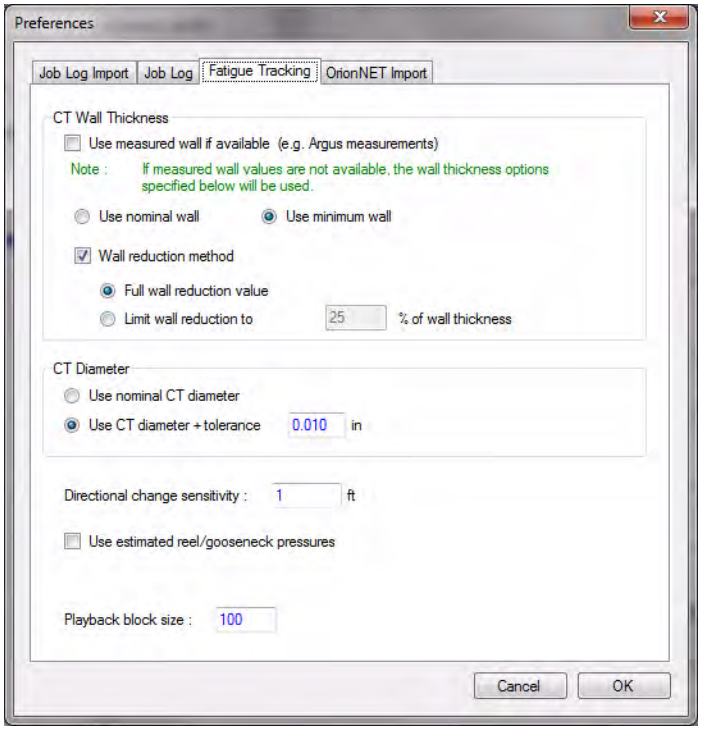
The information stored [in a database] during the Job Setup mode of the structured fatigue-tracking process is used to perform and store CT-string life calculations within the Fatigue Tracking mode. See "About Coiled Tubing (CT) Fatigue Tracking with Reel-Trak" on page 30 for additional information.

➢ To prepare to track fatigue

1. Open a job. (See "To set up a CT job" on page 34.)

To create a new job, the user may access the Job Wizard from the Reel-Trak toolbar .

2. From Reel-Trak's menu bar, select **Options** > **Preferences** to choose the various parameters used in the fatigue tracking calculations.



a. Select **CT Wall Thickness** parameters:

- Click to check the **Use measured wall if available** option if Argus TubeSpec™ measurement data is to be used.

Argus TubeSpec coiled tubing measurement data is imported automatically into Cerberus through Reel-Trak whenever an OrionNET™ database is selected and the above option is checked.

- Choose between **Use nominal wall** and **Use minimum wall**.
- To use the **Wall reduction method**, click (to check) the associated box; then, select **Full wall reduction value** or **Limit wall reduction to [enter percent] % of wall thickness**.

b. Select **CT Diameter** parameters:

- Choose between **Use nominal CT diameter** and **Use CT diameter + tolerance [use default or change number] [units]**.

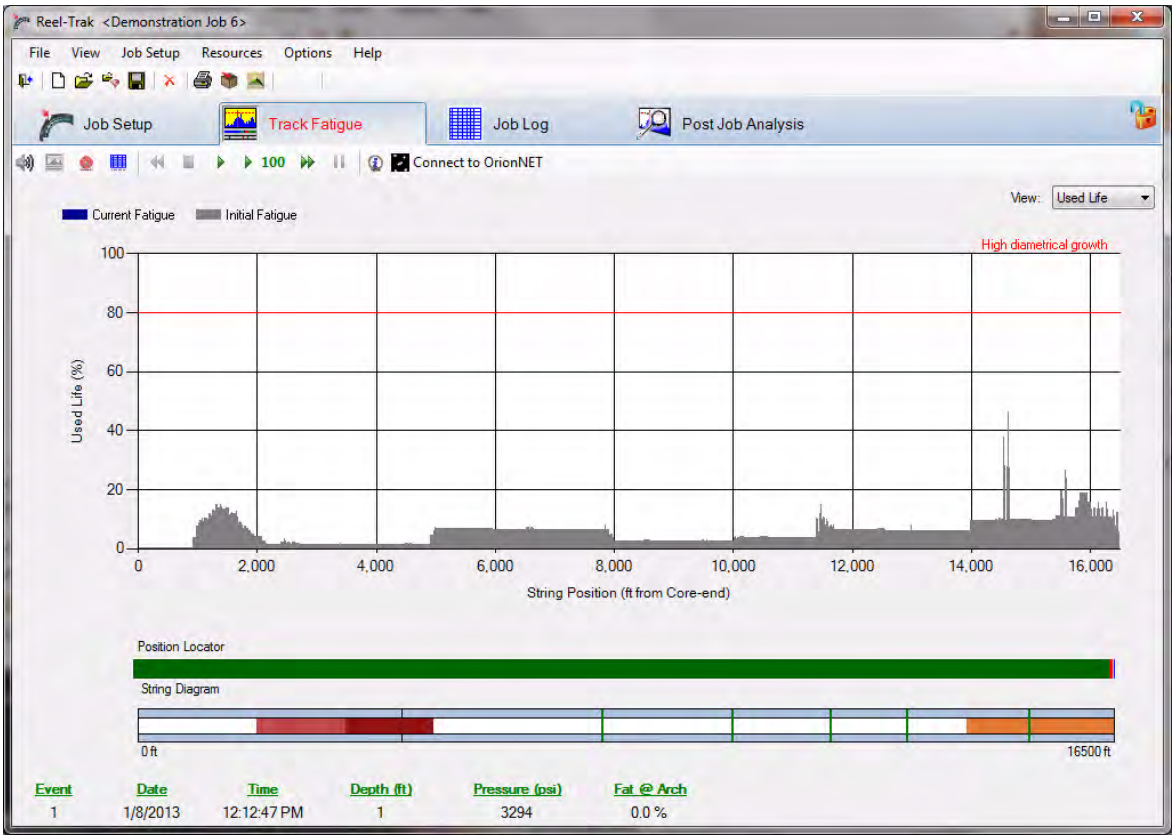
*In Reel-Trak, select **Options** > **Units** from the menu bar to change units.*

c. Specify other general parameters, if different from the default:

- **Directional change sensitivity: [use default or change number] [units]**.
- To **Use estimated reel/gooseneck pressures**, click to check the associated box.
- Use the default **Playback block size** or change the number within the associated field.

d. Click **OK** when selections and inputs are complete.

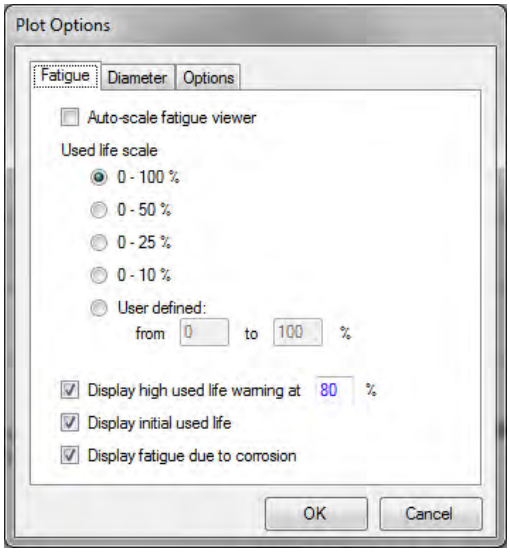
3. If not already selected, select the Track Fatigue tab in Reel-Trak.



Setting Up Fatigue Tracking Display Options

➢ To change the way the fatigue tracking plot displays

1. Right-click the plot area; then click **Options**. The Plot Options dialog appears. The Fatigue tab is the default.



 a. To choose scaling options, select from the available **Fatigue**/**Used life** options.

"Fatigue" and "used life" are interchangeable terms.

 b. To display a high used life warning, input scale percentage and click (to check) **Display high used life warning at [input] %**.

 c. To display initial fatigue (fatigue on the string before running the job), click (to check) **Display initial used life**. Initial used life appears in gray on the plot.

 d. To display fatigue due to corrosion, click (to check) **Display fatigue due to corrosion**. Corrosion fatigue displays in green on the plot.

Current or "new" fatigue appears in blue on the plot when the current job is running/run.

2. Select the Diameter tab to potentially display a high diameter warning. Choose between two high diameter display methods: percent growth or absolute growth.

 a. To display a high diameter warning, click (to check) **Display high diameter warning**.

 b. Then, click (to toggle) **Percent growth: Nominal diameter + [input] %** or **Absolute growth: nominal diameter + [input] [units]**.

3. Select the Options tab to select plot orientation, plot style, and various grid and string diagram options.

 a. Click (to toggle) **Plot orientation**: **Plot from Core-end** or **Plot from Free-end**.

 b. Click (to toggle) **Plot style**: **Line** or **Filled area**.

 c. To display the grid on the plot, click (to check) **Display grid**.

 d. To display the string diagram underneath the Position Locator, click (to check) **Display string diagram**.

e. To display effective zones in the string diagram, click (to check) **Display effective zones in the string diagram**.

Changes made to the string in String Editor (see "Using the String Editor" on page 215) are reflected in Reel-Trak. For example, effective zones and their derate percentage, when designated in String Editor, are reflected in Reel-Trak.

Understanding the Fatigue Diagram

When a job is open and the user accesses the Track Fatigue tab, the fatigue from previous job(s) run on the string are shown in gray in Reel-Trak's fatigue diagram. When fatigue is run for the current job, whether in real time (via OrionNET) or from recorded data, it appears in **blue**. Corrosion, if shown, appears in **green**.

Understanding the Position Locator

The **Position Locator** represents the current position (depth) of the reel and injector on the current string. This is used primarily for real-time fatigue tracking, but is also visible in playback mode. The following colors may be seen:

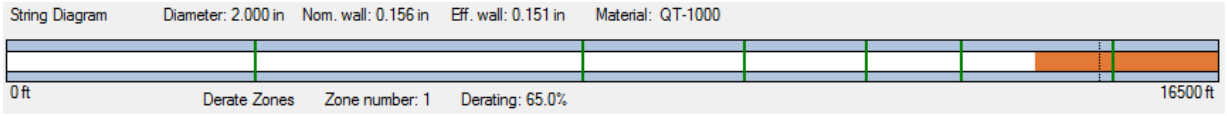
- **Green** = On the reel
- **Red** = Between the reel and the gooseneck
- **Light Blue** = From the first guide arch to the top of the injector
- **Blue** = Past the injector (into the well)



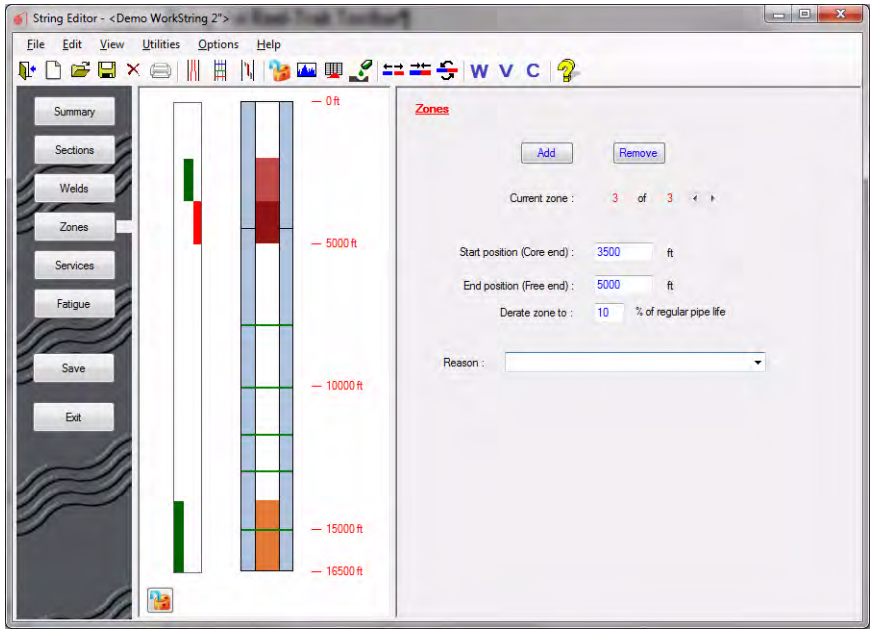
The coloration noted above, if present, may not be visible unless the user zooms in on the fatigue diagram. To zoom in, click and drag the mouse over the desired diagram area. To return the diagram to normal, right-click the diagram and select from the available size options.

Understanding the String Diagram

The **String Diagram** portion of the Track Fatigue tab shows vertical lines to indicate welds. Zones appear according to color, which is determined by the derate amount.



To set up the zones that appear in the **String Diagram**, access the String Editor (see "Using the String Editor" on page 215); then click the Zones tab. Note how the zone colors shown in the String Editor, below, correspond to those in If not already selected, select the Track Fatigue tab in Reel-Trak. above.



Running the Fatigue Job

The user may track fatigue in the following ways:

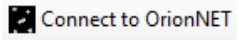
- In real time (via OrionNET).
- Post-job: After data is recorded in a database and fatigue has not yet been applied to the string.

When the job has been completed and fatigue applied to the string, the user can play back the fatigue-tracking process in playback mode.

Track Fatigue Toolbar

The following icons are visible and/or usable within the Track Fatigue toolbar.

Icon	Description
	Turn (toggle) alarm sound on/off.
	View string fatigue. This icon is greyed out in the Track Fatigue mode, but has color in the Job Log mode. From the Job Log mode, click this icon to move to the Track Fatigue mode.
	View Reel information.

Icon	Description
	View job log. From the Track Fatigue mode, click this icon to move to the Job Log mode.
	Reset fatigue tracking.
	Stop fatigue tracking (in real time).
	Run/apply one fatigue entry.
	Run/apply (n) fatigue entries (as set within the menu bar: **Options > Preferences > Playback Block Size**).
	Run/apply all fatigue entries.
	Pause fatigue tracking.
	Show diagnostic information.
	Connect to OrionNET.

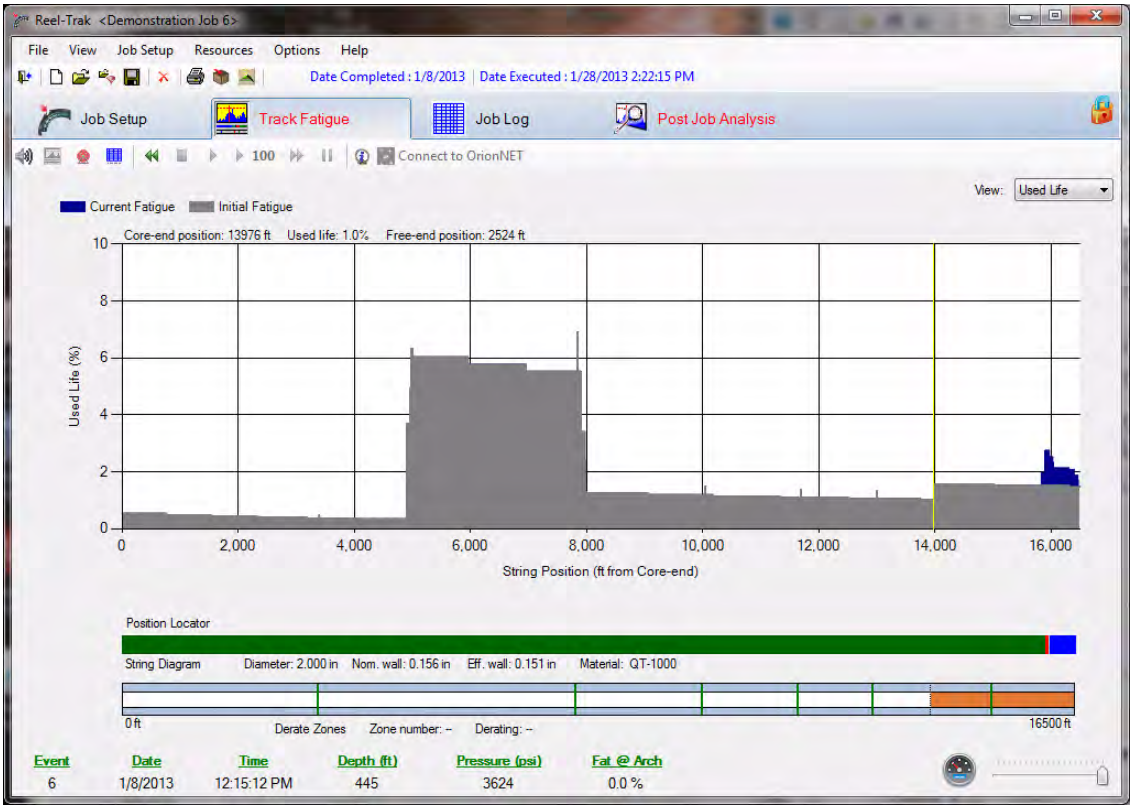
Running the Job from Recorded Data

When the job has not yet been run, but the data for the string exists in a database, the user can execute the fatigue tracking (run the job). The newly tracked fatigue shows on the grid in **blue**. If previous jobs have been run, that fatigue is considered “existing,” or “current,” and it appears on the grid in **gray**.

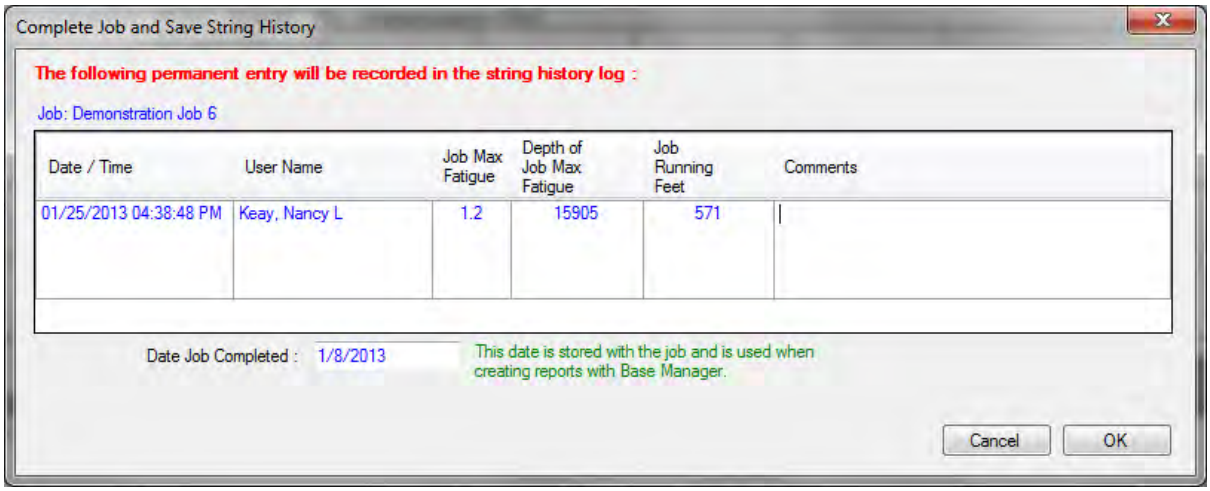
Use the job-execution controls described in Track Fatigue Toolbar  to run the job and to see the blue fatigue-tracking information appear in the grid.

The Position Locator shows the current position (depth) of the reel and injector on the current string (see “Understanding the Position Locator” on page 45 for additional information).

When the fatigue is run, the software applies the fatigue as it would have occurred at the time (in **blue**).



Save the job to permanently record the entry in the string history log.



This completes the job and applies all calculated fatigue to the string. The next time it is opened, Reel-Trak opens by default to the Post Job Analysis tab.

When the job is completed, the user may reset the job and run it in playback mode (see Running the Job in Playback Mode). The user may also “undo” the running of fatigue for a string via the String Editor (see “Printing String Data” on page 234). In this scenario, the tracked fatigue for the job is removed from the string and the job may be run as though fatigue had never been applied.

Running the Job in Real Time

➢ To run the job in real time

1. Click the **Connect to OrionNET** toolbar icon to start the process.

 Reel-Trak first checks to see if OrionNET is running on the computer and if a launch is needed. Reel-Trak then searches for the current database and connects to it when found, starting fatigue tracking.

 The user is prompted to select the current database if one cannot be identified.

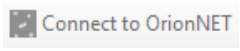
 Data imported into Cerberus is filtered, so the imported data may not correspond directly to the data in Orion-NET; only the entries required for fatigue tracking are imported.

2. When fatigue tracking is complete, click the **Save** icon on the general toolbar or the **Stop** button on the Track Fatigue toolbar.

3. Fatigue tracking stops; if the **Save** icon was selected, the software asks if the user wishes to apply all fatigue and complete the job.

 If the job is complete, a permanent entry is recorded in the string history log.

Running the Job in Playback Mode

If the job is complete, Reel-Trak opens in the Post Job Analysis tab/mode. The user may select the Track Fatigue tab and reset the job to run it in playback mode. The button to allow connecting to OrionNET becomes disabled 

, and a playback speed control becomes visible 
.

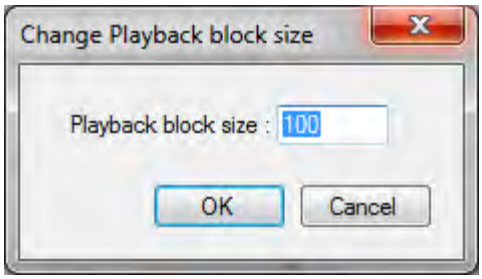
The fatigue for the string at the point prior to the job execution is loaded (appearing in **gray**). When the job is run in playback mode, the fatigue for the job (appearing in **blue**) is applied to the prior fatigue as it would have occurred during the job.

The playback block size may be set to determine how quickly playback occurs.

➢ To set the playback block size

1. Right-click the **Run** button 

to display the **Change Playback block size** dialog.



2. Input the desired block size and click **OK**.

Using the Post Job Analysis Tab

Four post-job functions are available in the Post Job Analysis tab/mode.

➢ To use the Post Job Analysis options

From the left-hand column, click the hyperlinks to view the associated grids.

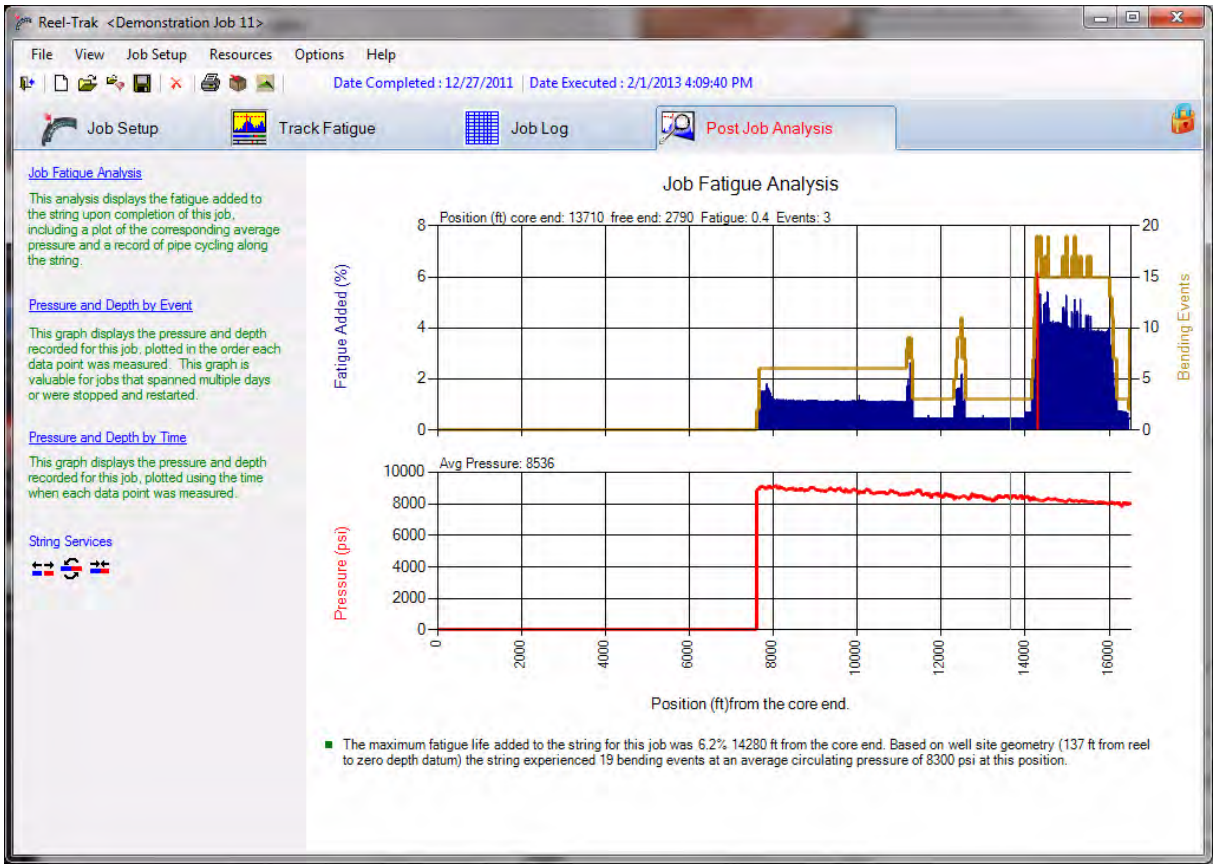
1. Job Fatigue Analysis (default)

 This plot represents an analysis of the fatigue added for the current job (not the total string fatigue, but the difference between the fatigue before the job and the fatigue after the job).

 The top plot renders this fatigue, as well as a single line representing the number of bending events that occurred at each depth during the course of the job. Additionally, the point of maximum fatigue is rendered in **red** to highlight it.

 The bottom plot depicts the average pressure at each depth over the job.

 The text at the bottom of the chart explains the maximum fatigue location and includes a reference to the wellsite geometry and average pressure. This analysis can be used to review and job and identify operations that use CT life more rapidly than others.

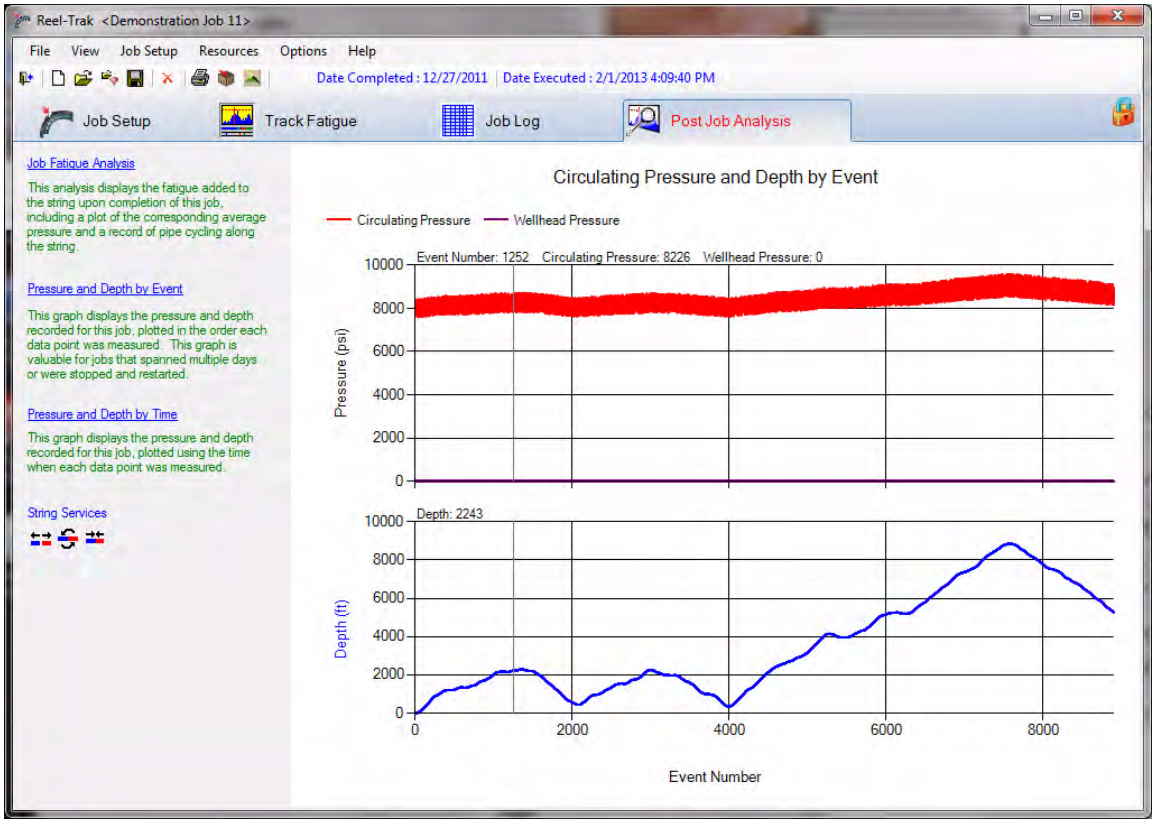


*Position is measured from the core end in the sample instance. The user may choose between depth being measured from the core end or from the end of the tool string, as described here: *Wellsite Geometry: The current default parameters for the Wellsite Geometry are displayed..*

Also described is the number of "bad" depth values filtered out of the data for proper viewing of the results (if the job was run in real time).

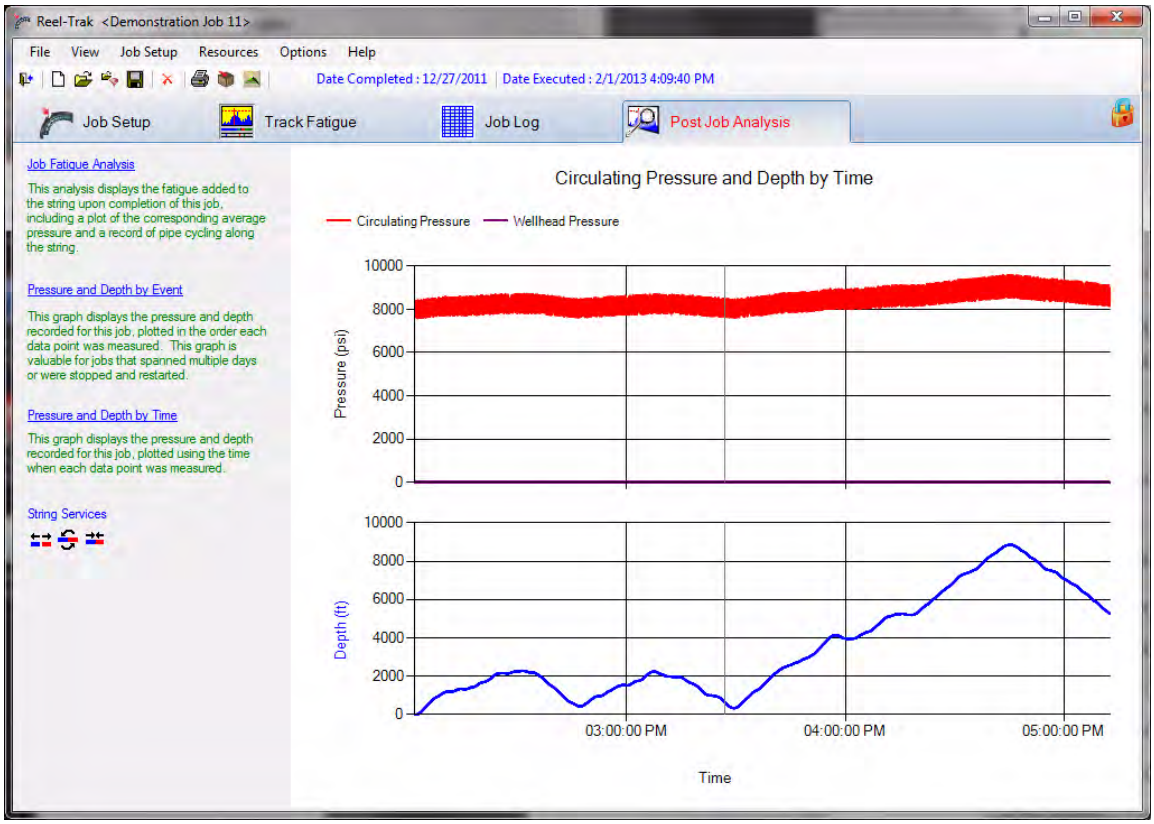
2. Pressure and Depth by Event [Number]

This graph depicts the pressure and depth by event number, independent of time. It can be used to review job activity when the job spans multiple days, or when there are large time gaps in the job log.



3. Pressure and Depth by Time

 This graph renders the pressure and depth by time. If there are any gaps in the data (for example, when the Data Acquisition System was not running or the data has been filtered), those gaps are rendered as a dashed line.



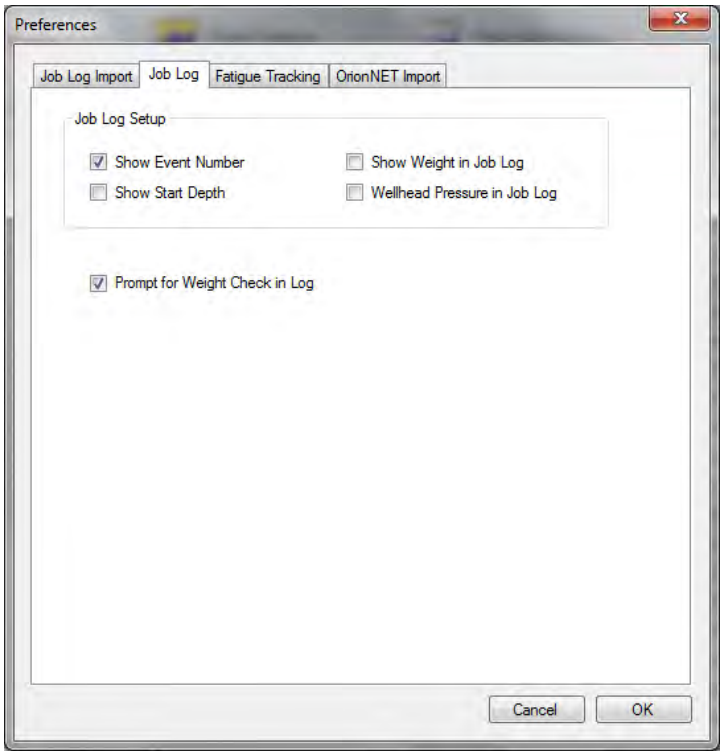
4. A String Services toolbar is provided for user convenience (see “String Services” on page 254 for more information).

Using the Job Log

Setting Up Job Log Display Options

➢ To change the way the job log displays

1. From the Reel-Trak menu-bar (when the Job Log tab is selected) click **Options > Preferences**. The Preferences dialog appears, with the Job Log tab appearing.



a. Choose **Job Log Setup** options.

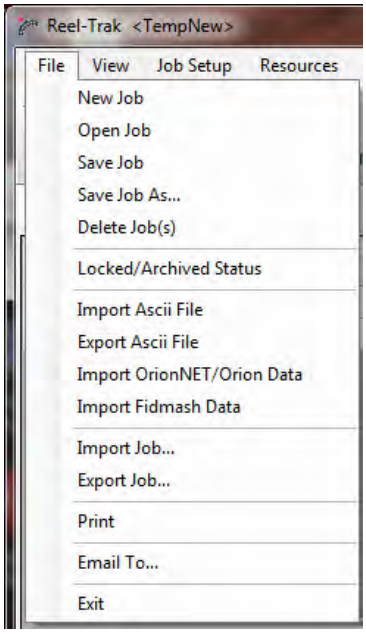
Job-log Toolbar Buttons

Job Log Toolbar

Tool Button	Tool Name	Function
	View String Fatigue	Moves to Track Fatigue tab in Reel-Trak.
	View Reel Information	Accesses Selected Reel dialog.
	View Job Log	Already accessed so grayed out.
	Import Orion/OrionNET Data	Accesses the Import Data dialog, from which the user may browse to select an Orion/OrionNET source file and select import options. Import options include data checks and the potential removal/replacement of “bad” or out-of-range data.
	Import ASCII file	Imports an ASCII text file into the current job log.
	Export Data	Exports the current job log as an ASCII file.
	New	Inserts a new row at the end of the job log.
	Insert	Inserts a row above the current row in the job log.
	Cut	Cuts the current selection and copies it to the clipboard.
	Copy	Copies the current selection to the clipboard.
	Paste	Pastes the contents of the clipboard at the insertion point.
	Start - Final Depth	Opens the Start and Final Depth dialog box.
	Running Feet	Opens the Running Feet dialog box to calculates running feet for the current job log.

Job-log Menu-bar Selections

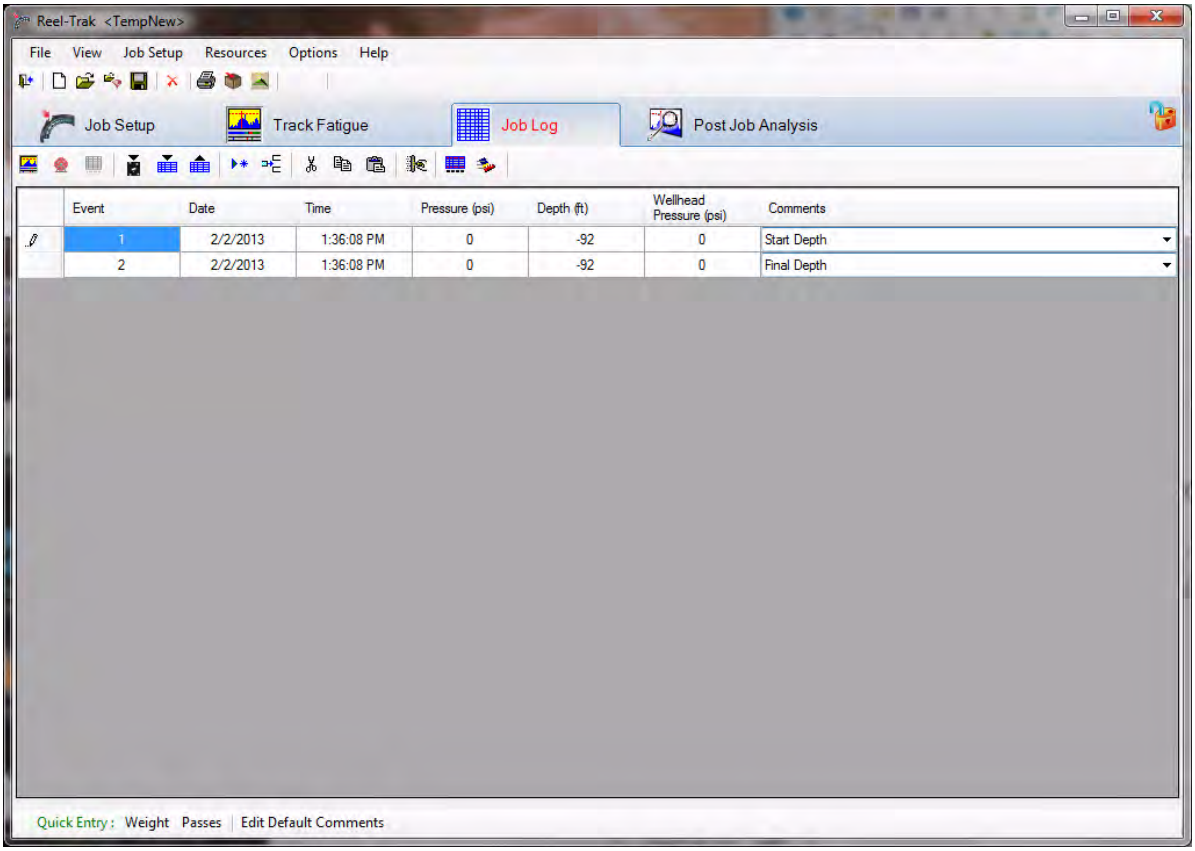
When the Reel-Trak Job Log tab is selected, use the Reel-Trak **File** menu-bar options to complete some of the import/export actions described in Job-log Toolbar Buttons.:



2. Configure the job-log data.

 Use the available toolbar buttons and/or menu-bar selections to assist in the job-data configuration.

Job Log Job-data Display Screen before Data is Entered/Imported



Each row of job data represents one “event.” A row should be entered at least each time directions change (such as when checking weight), and each time the pressure changes significantly. To enter data in a row, simply type within the table.

The following table describes the columns in the Job Data tab within the Job Log.

Column	Required?	Description
Event Number	Yes; automatically generated.	The rows are automatically numbered sequentially.
Date	Yes, for creating charts in the Job Log.	Date at the end of the event.
Time	Yes, for creating charts in the Job Log.	Time at the end of the event.
Pressure	Yes	The circulating pressure is readily available and yields a conservative fatigue prediction.
Depth	Yes	Enter the depth

Column	Required?	Description
Wellhead Pressure	No	Enter (or, more likely, import from an Orion file) the actual wellhead pressure (psi) throughout the job rather then having to use an average value for the entire job.
Comments	Yes; automatically generated.	Enter a comment, or select from available comments for record-keeping purposes.

- To enter a Check Weight row within the job-data table

 1. Select the row immediately before the row where “check weight” should appear.
 2. Click **Weight** at the bottom of the display screen.
 3. Enter the **Pressure**, **Pull up amount**, and **comments**, or use the defaults.

- To enter Multiple Passes within the job-data table

 1. Select the **row** immediately before the row where the passes should appear.
 2. Click **Passes** on the bottom of the screen.

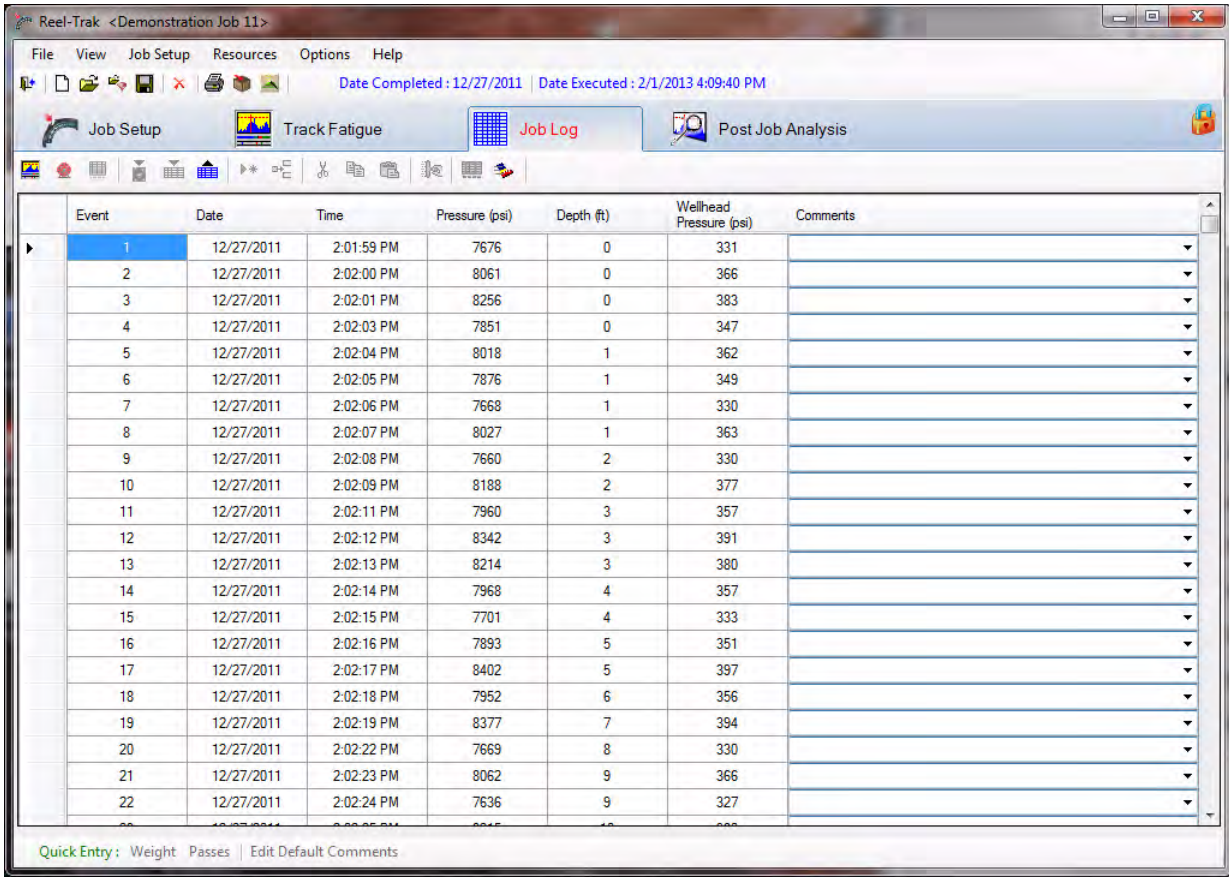
- To edit the Default Comments available within the job-data table

 1. Click **Edit Default Comments** on the bottom of the screen.
 2. Use the available buttons to add, delete, or save comments from which to select in the screen dropdown.



When real-time fatigue tracking has started, all become disabled with the exception of the Export to ASCII and Calculate Running Feet functions.

Job Log Post-Job-Run



Event	Date	Time	Pressure (psi)	Depth (ft)	Wellhead Pressure (psi)	Comments
1	12/27/2011	2:01:59 PM	7676	0	331	
2	12/27/2011	2:02:00 PM	8061	0	366	
3	12/27/2011	2:02:01 PM	8256	0	383	
4	12/27/2011	2:02:03 PM	7851	0	347	
5	12/27/2011	2:02:04 PM	8018	1	362	
6	12/27/2011	2:02:05 PM	7876	1	349	
7	12/27/2011	2:02:06 PM	7668	1	330	
8	12/27/2011	2:02:07 PM	8027	1	363	
9	12/27/2011	2:02:08 PM	7660	2	330	
10	12/27/2011	2:02:09 PM	8188	2	377	
11	12/27/2011	2:02:11 PM	7960	3	357	
12	12/27/2011	2:02:12 PM	8342	3	391	
13	12/27/2011	2:02:13 PM	8214	3	380	
14	12/27/2011	2:02:14 PM	7968	4	357	
15	12/27/2011	2:02:15 PM	7701	4	333	
16	12/27/2011	2:02:16 PM	7893	5	351	
17	12/27/2011	2:02:17 PM	8402	5	397	
18	12/27/2011	2:02:18 PM	7952	6	356	
19	12/27/2011	2:02:19 PM	8377	7	394	
20	12/27/2011	2:02:22 PM	7669	8	330	
21	12/27/2011	2:02:23 PM	8062	9	366	
22	12/27/2011	2:02:24 PM	7636	9	327	

Packaging a Job

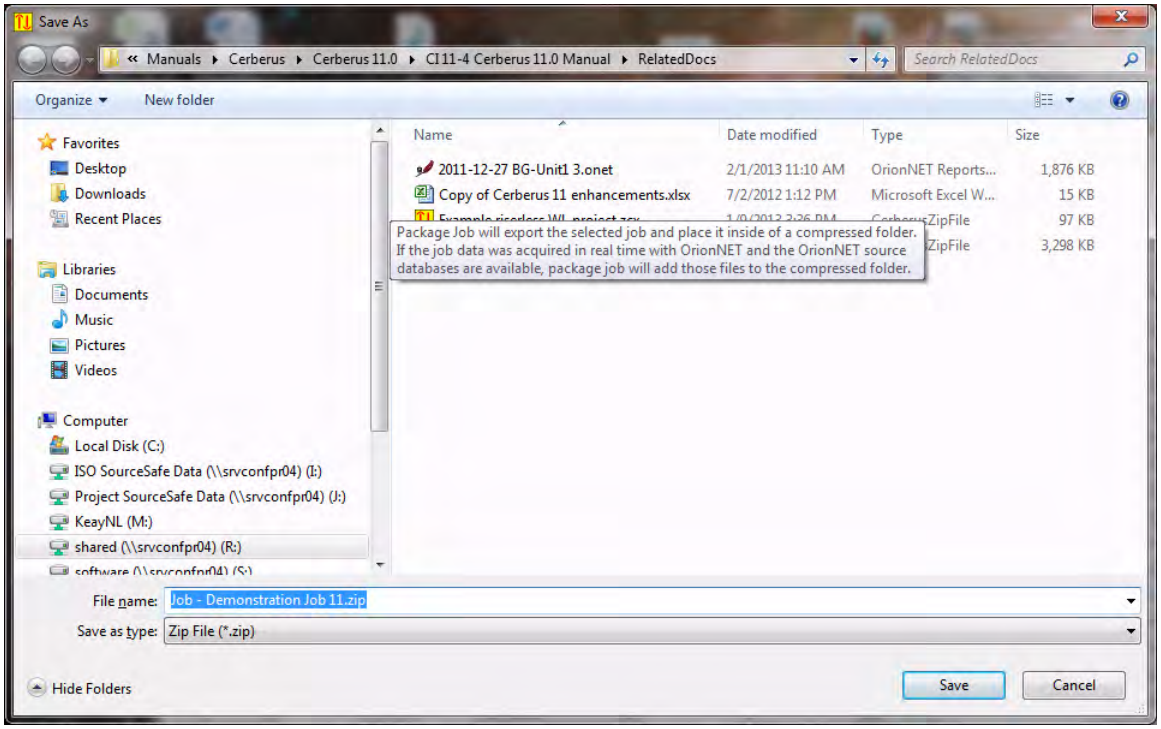
Reel-Trak can package a job and place it inside of a compressed folder. If the job data was acquired in real time with OrionNET and the OrionNET source databases are available, Package Job adds those files to the compressed folder.

Package Job allows the reproduction of all of the fatigue tracking to occur at the base after the job is complete.

➢ To package a job

1. Click the **Package Job** icon from the Reel-Trak menu bar.

 The Save As Windows-like dialog appears.



2. Name the file and click **Save**.

Chapter 5
Using Orpheus

This section describes the process involved in using Orpheus [Well Intervention Modeling] – the Cerberus forces model.

About Orpheus

When determining the feasibility of a job, it is essential to calculate the cumulative mechanical and hydraulic forces likely to affect conveyance methods (coiled tubing, wireline, slickline, or jointed pipe).

Orpheus calculates these cumulative forces, providing immediate feedback so that the user can resolve issues before they become insurmountable problems.



See "Coiled Tubing Forces Theory" on page 389 for information concerning forces theory for coiled tubing.

The Orpheus Process

The process involved in using Orpheus is as follows:

1. Create a new Orpheus project or select an existing project (see page 64).
2. Determine a conveyance method (see page 69).
3. Create a tool string or select/edit an existing tool string (see page 72).
4. Create a well or select/edit an existing well (see page 73).
5. Create a fluids profile or select an existing profile (see page 73).
6. Set calculation parameters (see page 77).
7. Perform calculations (see page 85).
8. Print results for hardcopy or electronic transmission (see page 131).

Accessing Orpheus

➢ To access Orpheus from the Cerberus home page

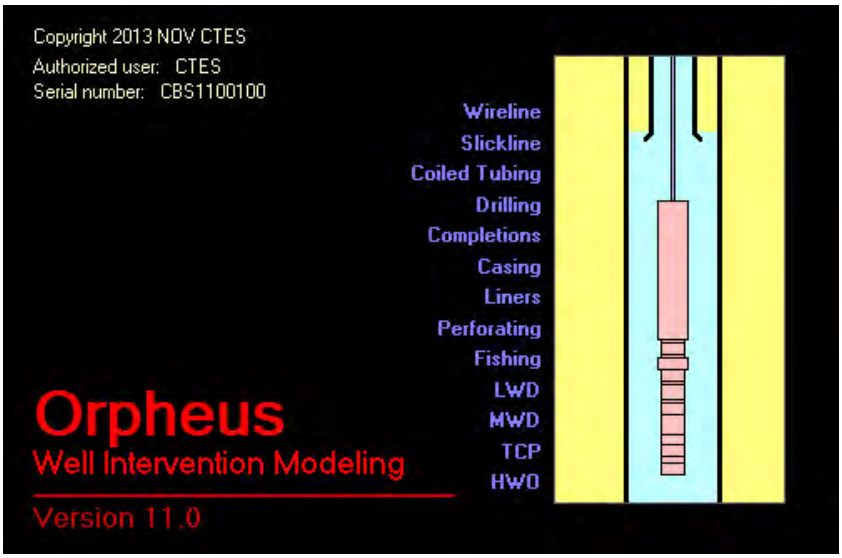
- Select the Orpheus button.



OR

- Select **Models** > **Orpheus** from the menu bar.

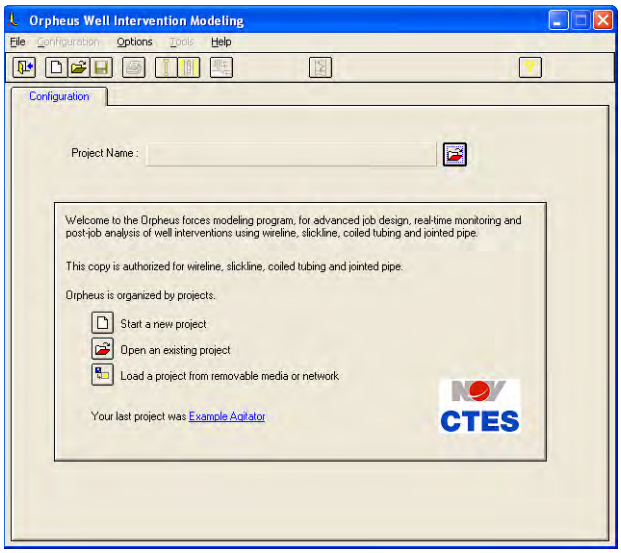
 The Orpheus model loads, as indicated by the following display screen:



As noted within the opening dialog, Orpheus (along with the other Cerberus models) is organized by projects.

From the Orpheus Well Intervention Modeling dialog, choose to start a new project, open an existing project, or load a project from removable media or a network:

Orpheus Opening Dialog



Working with Projects

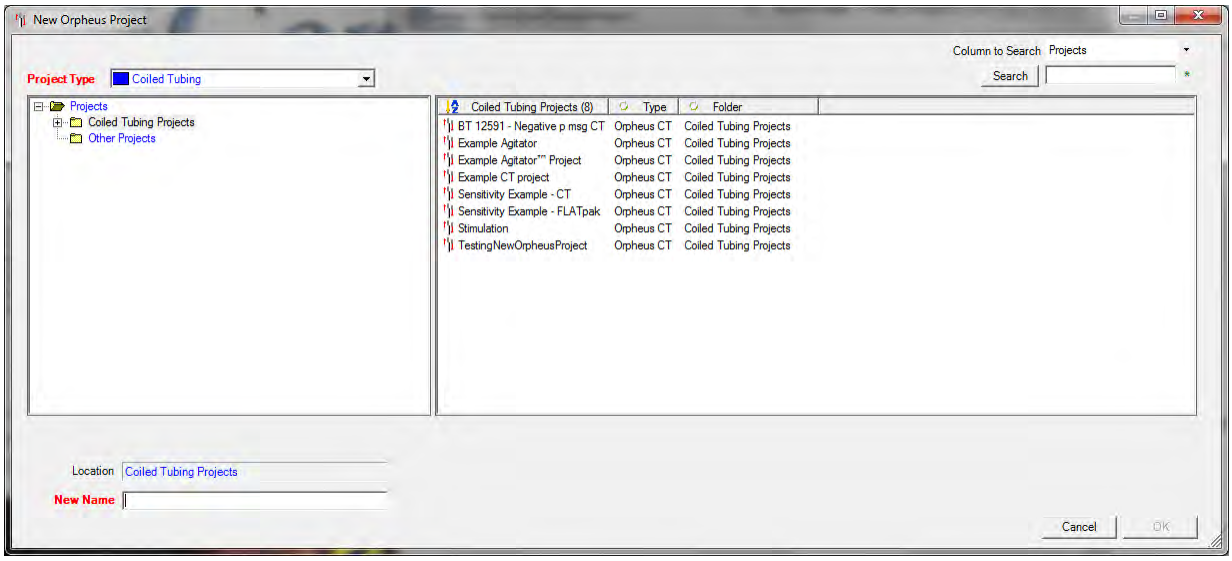
Creating a New Orpheus Project

The following procedure outlines how to create a new Orpheus project.

➢ To start a new project

1. **From the Orpheus opening dialog**, choose one of the following:
 - Select **File** > **New Project** from the menu bar.
 - Click the **Start a new project** button on the Configuration tab.
 - Click the **New** toolbar button.
 - Select **Help** > **Orpheus Wizard** from the menu bar.
 The Orpheus Wizard is a simplified Orpheus-project creation tool. The Orpheus Wizard becomes available only after creating a name for a new project. See New Project Dialog for additional information.
 - The **New Orpheus Project** dialog appears:

New Project Dialog



Start a new Orpheus project directly from the Cerberus home page by selecting **File > New > Forces Project (Orpheus)**

2. Select the **Project Type** to use in creating the project.

Only the project types (Coiled Tubing, Wireline, or Jointed Pipe) with existing licenses are visible/available.

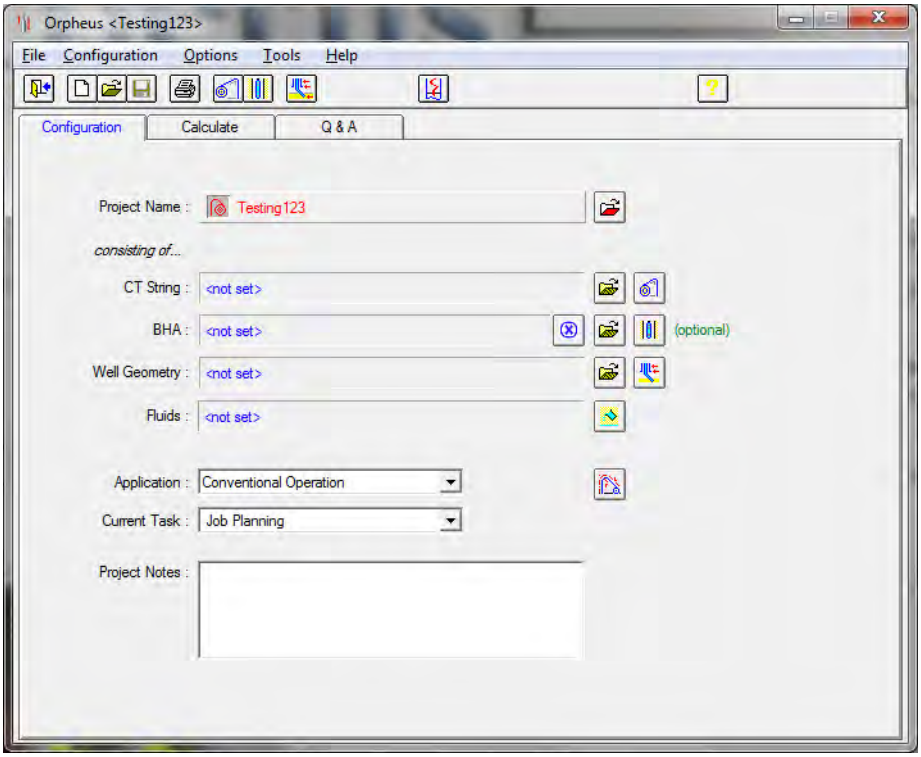
Modify the ***licensed*** *project types' availability for viewing by selecting* **Options > Orpheus Preferences** *from the main Orpheus display screen.*

3. Search for existing projects using the **Column to Search** dropdown and the direct-input field in the upper right-hand corner of the New Orpheus Project dialog. Click **Search** to carry out the search operation.

4. Provide a **New Name** for the project. The name may consist of letters, numbers, and spaces, with a maximum of 32 characters and click **OK**.

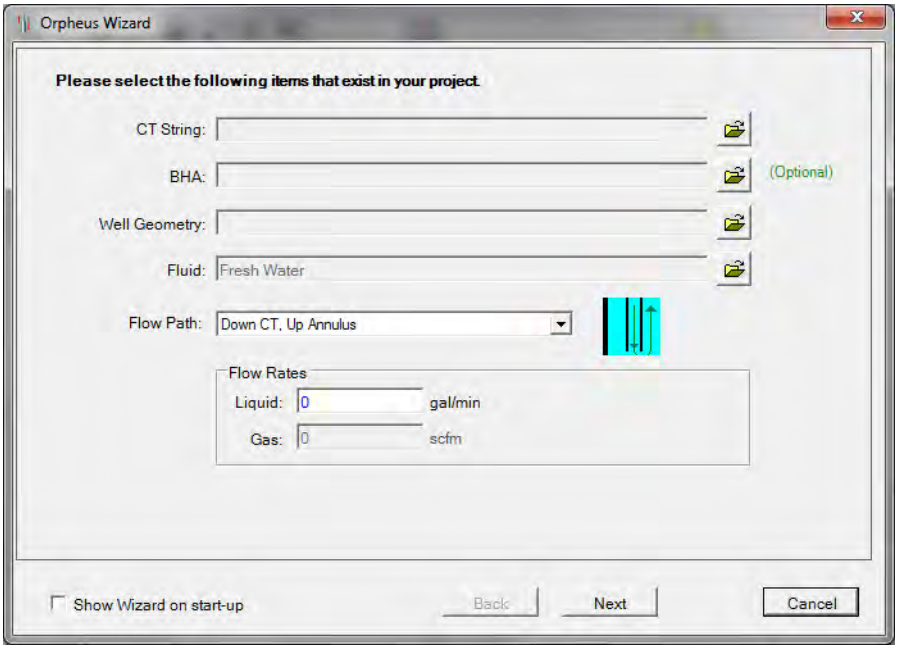
 One of two things happens:

a. The project-description dialog appears.



OR

The Orpheus Wizard dialog appears.



a. The Orpheus Wizard dialog appears only when one of the following circumstances applies:

- The New Orpheus Project dialog is accessed via **Help > Orpheus Wizard** from the Orpheus menu bar.

 OR

- The **Show Wizard on start-up field** was clicked (checked) when the wizard was open previously.

5. Configure the project using either the wizard or the standard project-creation interface (see Entering Project-configuration Information for more information about configuring a project using the standard interface).

Opening an Existing Orpheus Project

The following procedure outlines how to open an existing project.

➢ To open an existing project

1. Start Orpheus.

1. **From the Orpheus opening dialog**, choose one of the following:
 - Select **File > Open...** from the menu bar.
 - Click the **Open an existing project** button  on the Configuration tab.
 - Click the **Open** toolbar button. 

2. The Select an Orpheus Project display screen appears.

3. Select the project-type folder.



The folders associated with active modes/project types are available.

4. Click the name of the project.

Organizing Projects into Folders

Orpheus is designed with default folders for each type of project (Coiled Tubing Projects, Jointed Pipe Projects, and Wireline Projects). The default folders cannot be renamed, moved, or deleted.

When a new project is created, a folder for the project's placement is automatically selected.



These folders are internal to Cerberus, and unrelated to folders in Windows Explorer.



Projects or folders cannot be placed under different project types. (For example, CT projects cannot be placed under the Wireline Projects folder.)

Saving an Orpheus Project

The project configuration and last set of calculated values are retained when the project is saved. There are two ways a project can be saved:

- Click **Save** () on the toolbar.

OR

- Click **File** and select **Save** or **Save As...**.

Deleting an Orpheus Project

Unnecessary projects can be deleted. When a project is deleted, the information is not recoverable.

➢ To Delete a Project

1. Click **File** > **Delete**.

 The Select Orpheus display screen appears.

2. Select the folder that contains the project.

3. Click on the project name.

4. Click **OK**.

 The **Delete Project** warning box appears to confirm deleting the selected project.

5. Click **Yes**.

 The selected project is deleted and the user is returned to the Orpheus main display screen.

Importing an Orpheus Project

Any exported project can be imported into Orpheus.

➢ To import a project

1. If importing from removable media, insert the media into the PC.

2. Start **Orpheus**.

3. Select **File** > **Import Project...** from the dialog menu bar.

 The Select an Orpheus project to import dialog box appears.

4. Select the **project(s)** to import.

5. Click **Open**.

 Orpheus imports the project data.

6. Click **Yes** to open the project.

Exporting an Orpheus Project

Any project in Orpheus can be exported to removable media to be imported into another computer with Orpheus.

➢ To Export a Project

1. If exporting to removable media, insert the media into the PC.
2. Start **Orpheus**.
3. Click **File** > **Export Project...** from the dialog menu bar.

 If a project is not already selected, the Select Orpheus Projects dialog appears. If a project is selected, the Export Orpheus Project dialog box appears.
4. From the Select Orpheus Projects dialog, select the project-type folder.

 This is the project-type folder with the desired project in it.
5. Select the project(s) to export.
6. Click **OK**.

 The Save As dialog box appears.
7. Navigate to the desired export location.
8. If desired, enter a new name for the project.

 If the project is to retain the current name, proceed to Step 9. Otherwise, enter a new name.

 Ensure the file type is Cerberus File (*.zcx).
9. Click **Save**.
10. Click **OK**.

 The project is exported to the desired location and file and the user is returned to the Orpheus main screen.

Entering Project-configuration Information

Configuration information is stored within a project, so it is quickly accessible without the necessity of re-input.

➢ To enter Coiled Tubing (CT), Jointed Pipe (JP), or Wireline (WL) project-configuration information

1. Open an existing Orpheus project or create a new project.
2. Use/edit an existing conveyance string/cable or create a new string/cable.

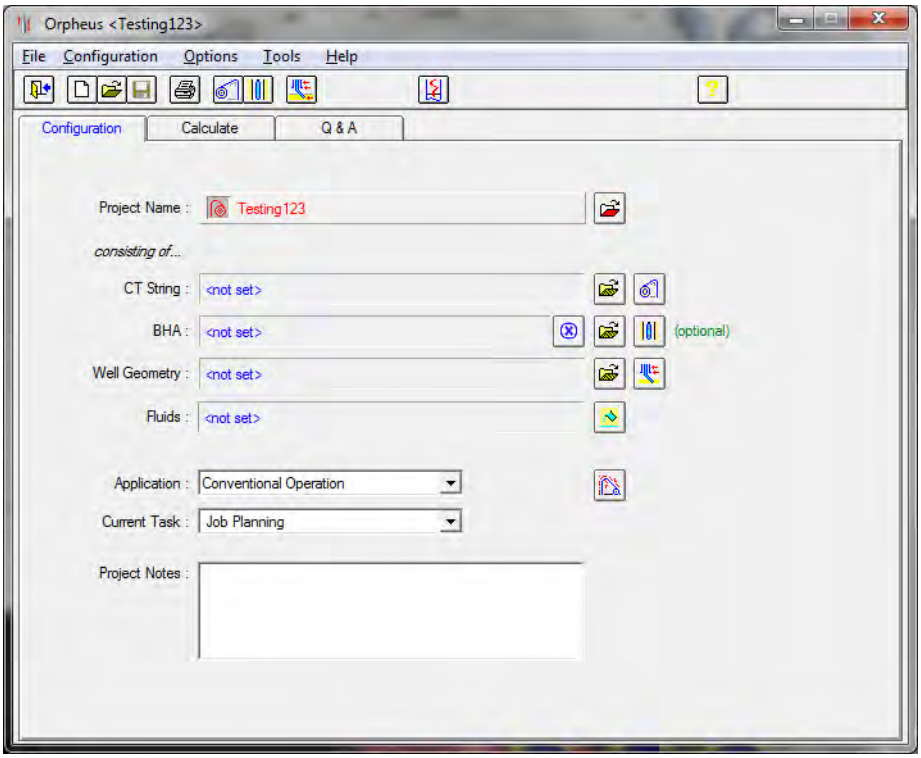
 Click the yellow-folder icon to browse among conveyances and select from existing CT Strings, JP Strings, or WL cables.

 OR

 Choose another icon associated with the conveyance string/cable:

a. For a Coiled Tubing Project

Orpheus Coiled Tubing Configuration Tab

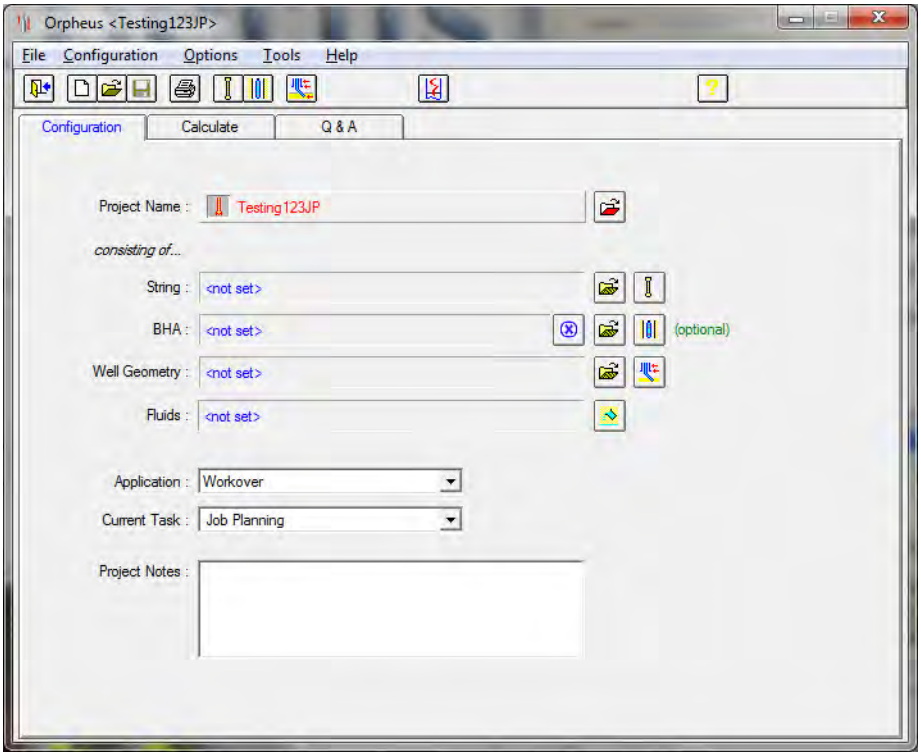


Click the Edit String () icon to access the String Editor display screen. Then, select **File > Open** from the menu bar. The Select a String dialog appears. Here, select an existing string.

If the CT String selected is of type FLATpak™, hydraulics and catastrophic buckling calculations are not available with this release of Cerberus.

b. For a Jointed Pipe Project

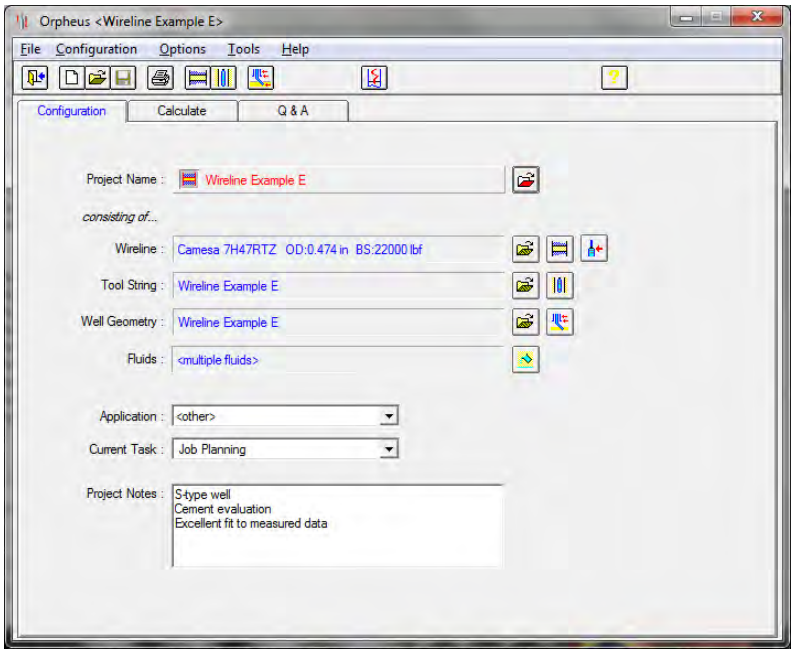
Orpheus Jointed Pipe Configuration Tab



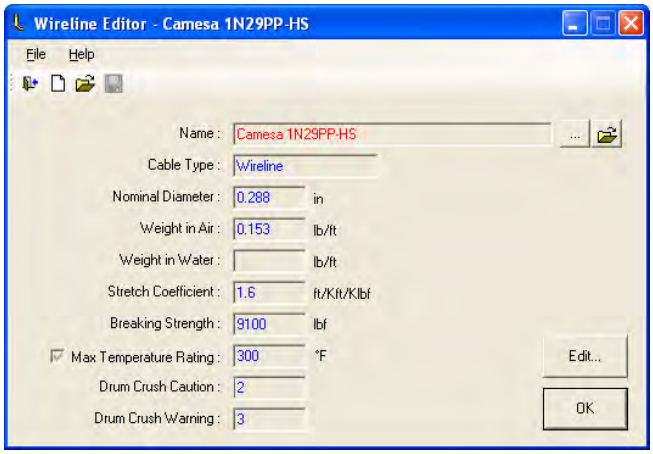
Click the Edit Pipe () icon to access the Jointed Pipe Editor display screen. Then, select **File > Open** from the menu bar. The Select Pipe dialog appears. Here, select an existing pipe.

c. For a Wireline Project

Orpheus Wireline Configuration Tab



Click the Edit Cable () icon to select a wireline if none is selected and to access the Wireline Editor dialog:



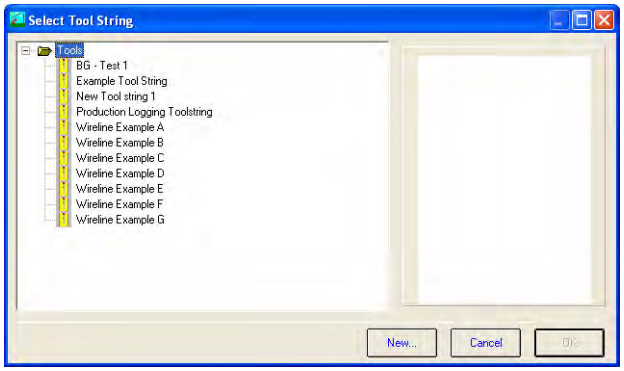
Then, click the ellipses to select from recently used cables, or click the yellow-folder icon to access the Select Cable dialog, from which another cable may be selected.

To edit the wireline weakpoint, click the Edit Weakpoint () icon to access the Weakpoint Project Options dialog.

3. (Optional for CT and JP) Use/edit an existing Bottomhole Assembly (BHA)/toolstring or create a new BHA/toolstring.

 Click the associated yellow-folder icon to browse and select from existing BHAs/toolstrings.

OR click the Tool String () toolbar icon to access the Select Tool String dialog. Here, select an existing toolstring or click **New** to create a new toolstring.



4. Use/edit existing well or create a new well.

 Click the Well Geometry yellow-folder icon to browse and select from existing wells.

 OR click the Edit Well () toolbar icon to access the Well Editor display screen. Here, select File > Open from the menu bar to access the Select a well dialog, from which the user may select an existing well.

5. Select fluids for use in the project.

 Click the Select Fluids icon () to access the Fluid Distribution dialog.

 For details, see “Fluid configuration” on page 74.

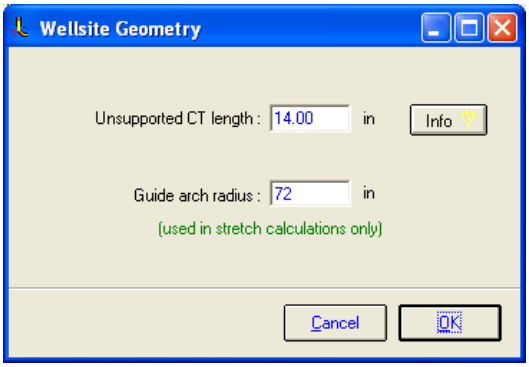
6. Select the job **Application** from the available dropdown. The selected application controls the available inputs on the Calculate tab:

 For Coiled Tubing: Conventional Operation or Coiled Tubing Drilling determine the allowed user inputs in the software; Other allows all inputs.

 For Wireline: Open Hole Operations, Cased Hole Operations, or Other.

 For Jointed Pipe: Workover, Drilling (Conventional), Drilling (Underbalanced), Hydraulic Workover (HWO), Pipe Conveyed Logging (PCL), Tubing Conveyed Perforating (TCP), Running Completion, Running Casing, Running Liner, or Other.

a. Select the **Wellsite Geometry** icon (CT) – adjacent to the **Application** field () – to input Unsupported CT length and Guide arch radius from the related dialog. This information is important for the catastrophic buckling calculation and the stretch calculations.



7. Select the **Current Task** from the dropdown: Job Planning, Real-time Job Monitoring, Post-job Analysis.

8. Add **Project Notes** as desired.

9. **Fluid configuration**

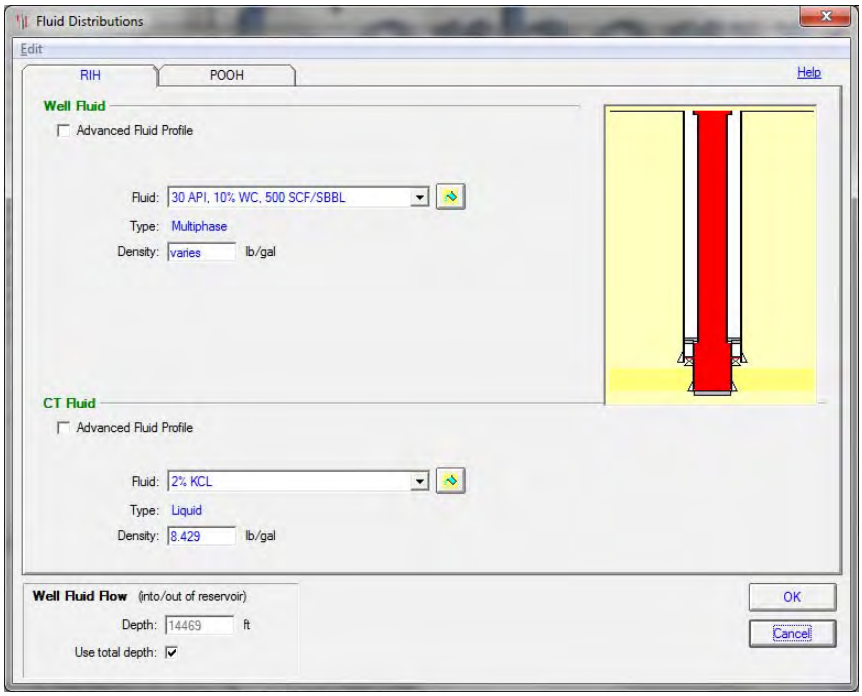
 a. Fluid Distribution

 Launch the Fluid Distribution display screen using one of two methods:

 - From the Configuration tab, click the Select Fluids icon .

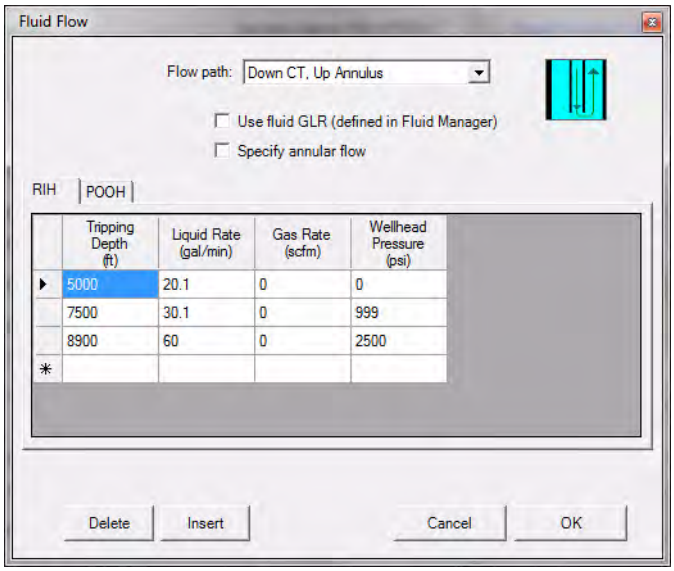
 OR

 - From the Calculate tab, click the button adjacent to the fluid density fields (when the conveyance is wireline, no button for CT/JP fluid density appears).



b. Fluid Flow

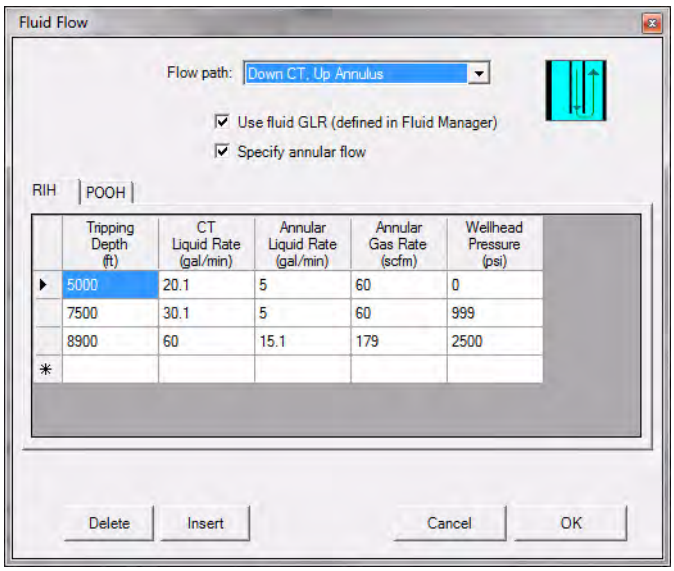
Click the icon adjacent to the flow-rate field(s) to reveal the Fluid Flow dialog.



Edit the Fluid Flow table(s):

- Within the table, select rows by clicking within the far-left-hand column, and use the **Delete** and/or **Insert** buttons to delete or insert rows.
- Input data.
- Select the RIH and POOH tabs to input data for both.

Two options are available:



For multi-phase fluids, the user has the option to apply the GLR defined in the fluid manager to the liquid flow rate and the gas flow rate. See Gas/Liquid Ratio (GLR).

The user has the option to define annular flow rates differently from the CT/JP flow rates. See Annular Flow.

c. Gas/Liquid Ratio (GLR)

For multi-phase fluids, the user has the option to apply the GLR defined in the fluid manager to the liquid flow rate and the gas flow rate.

- If the user clicked (to check) the **Use fluid GLR (defined in Fluid Manager)** togglebox (available only if a multiphase fluid is used), then the gas flow rate is automatically calculated based on the user-entered liquid flow rate and GLR value (or the liquid flow rate is automatically calculated based on the user-entered gas-flow rate and GLR value).
- If the user does not select **Use fluid GLR (Defined in Fluid Manager)**, then both liquid flow rate and gas flow rate may be input separately.

*When the **Use fluid GLR (defined in Fluid Manager)** is selected, the GLR button's background color changes to light yellow. Hold the mouse over the button to see the tool tip.*

d. Annular Flow

The user has the option to define annular flow rates differently from the CT/JP flow rates.

- If **Specify annular flow** is selected, then both liquid flow rate and gas flow rate are defined (input) separately.
- If **Specify annular flow** is not selected, then the flow rates apply to both the CT/JP and the annulus.

*When the **Specify annual flow** is selected, the related button's background color changes to light yellow. Hold the mouse over the button to see the tool tip.*

Entering Project Options

Project-option settings provide input for, and affect the outcomes of calculations, so it's important to enter these accurately.

➢ To enter project options

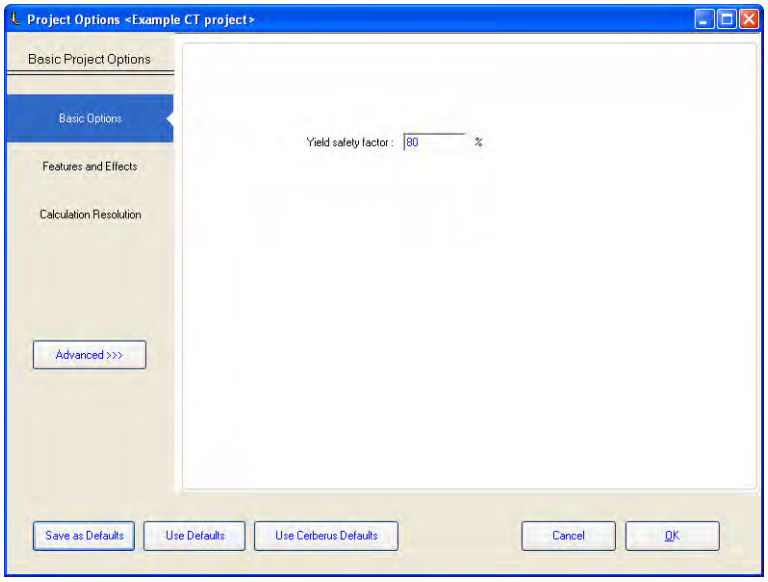
Access project options one of two ways:

1. Select **Options > Project Options** from the Orpheus menu bar when a project is open.

 OR

 Select the Calculate tab from the Orpheus project display screen; then, click the **Project Calculation Options** button in the upper far-right-hand panel.

2. The Project Options display screen appears. This screen is divided into three separate areas: left panel, right panel, and bottom panel.



 The Project Options display screen defaults to the Basic Project Options view (see the extreme top portion of the left panel in the above figure). Here, within the left-hand panel, the user sees three tab-like, selectable choices: Basic Options (default), Features and Effects, and Calculation Resolution.

3. From the default Basic Options tab, input the following:

 - Yield safety factor (CT, JP)

 Coiled tubing or jointed pipe is considered to be at its working limit when the stress (axial stress or von-Mises stress) at any point in the string reaches this percentage of the material yield strength. Yield strength is a property of the steel and is usually expressed in kpsi. The yield safety factor is 80% by default. Modifying this value causes all calculations in Orpheus that are limited by yield strength of the string to be changed. An example is the "Yield limit" curve on the Surface Weight graph of the Trip in and Out calculation.

 - Safe Operating Load (% of breaking strength) (WL)

 Wireline is considered to be at its operating limit when the tension at any point in the cable reaches this percentage of the cable breaking strength. The safe operating load is 50% by default. Modifying

this value causes all calculations in Orpheus that are limited by breaking strength of the cable to be changed. An example is the “Safe Operating Load” curve on the Surface Weight graph of the Run In and Out calculation.

- Safety margin for tractor pull (WL)

 This is used in calculating the minimum required tractor pull to help the toolstring run in-hole. Theoretically, required tractor pull is the minimum pull that ensures the entire length of the wireline is in tension. However, sometimes it is preferable to keep the tension above a minimum positive value. Safety margin for tractor pull is this threshold and should not be less than zero.

 Note the difference between the required pull and the required pull with safety margin usually is not just the value of the safety margin. Instead, the program uses the safety margin value as the threshold for wireline compression. In other words, the program searches for the required minimum pull force so that the tension along the wireline is not less than the safety margin value.

- Real Axial Force Monitoring (JP)

 - Enable real axial force monitoring (JP)
 - Distance from end of string (JP)

 These two options, working together, allow the user to monitor real axial force at a specific position from the bottom of the string. Note the word “real” does not mean “field-measured” forces (i.e., weight indicator readings), but rather the axial force including the effect of any fluid pressure. When the option is selected and the distance from end is specified (say, xxx ft), the graph window displaying the Trip In and Out results will contain a graph called “Real Axial Force at xxx ft from End of String.”

4. Select Features and Effects; click (to check) any of the following to activate:

 - Include Disconnect Limit (CT)

 Though intended primarily for wireline mode, users can utilize this feature to model the CT connector and/or shear/pin type disconnects. When enabled, surface weight and the connector tension corresponding to disconnect limits can be calculated from the graph-window display for Trip In and Out.

 - Include Weak Point (WL)

 Though intended primarily for wireline mode, users can utilize this feature to model the CT connector and/or shear/pin type disconnects. When enabled, surface weight and cablehead tension corresponding to weak point limits can be calculated from the graph-window display for Trip In and Out.

 - Include ‘Buckled Production Tubing’ Effect (CT, JP, WL)

 By selecting this option, the user is able to simulate intervention in buckled production tubing. In order to include the effect of buckled production tubing in Orpheus calculations, the user must first run the Packer and Completion Analysis and then use the Tool Passage analysis (inside the Packer and Completion Analysis) to update the Orpheus well with the buckled production tubing configuration.

 - Include Additional Micro Tortuosity (CT, JP, WL)

 Select this option to include the additional frictional force caused by the micro-tortuosity (or spiral tortuosity). The force calculation for a pipe or cable section in a spiral wellbore is the same as that for coiled tubing in a helically-buckled production tubing.

 Micro-tortuosity is defined as the tortuosity that occurs on a much smaller scale as compared to the tortuosity (or macro-tortuosity) that represents the difference in dogleg severity between the planned

well survey and the final well survey. Therefore, micro tortuosity will not normally be observed in conventional survey data. The primary source of micro tortuosity is borehole spiraling, where the borehole axis resembles a helix instead of a straight line.

- Include Jar Activation Calculation (WL)

 Select this option to enable the calculation of surface weight and cablehead tension corresponding to low-setting and high-setting jar firing forces, respectively. The option is available only if the toolstring contains a jar.

- Include Downhole Agitator™ (CT, JP)

 Select this option to include friction mitigation due to the gentle pulsation of the pipe provided by the presence of a downhole pulsation device (like the NOV Downhole Agitator™ tool). The friction mitigation effect gradually attenuates along the length of the CT/JP string and a point is reached where there is no significant friction mitigation effect. The model used for this analysis finds that point, and from that point upward the friction is calculated and applied as it would be without the presence of an Agitator tool.

 The user provides the value of the ratio of friction drag reduction to flow rate, which (together with the current flow rate in the CT/JP string) is used to determine the total amount of friction force that the Agitator tool should remove. At a flow rate of 120 gal/min, a ratio of 13 lbf/(gal/min) yields 1,560 lbf of friction removed, which agrees with experience documented in the technical paper of SPE 121752.

- Include Additional Normal Force (WL)
- Include Tubing Collapse (CT, JP) - ellipses button launches collapse settings form.

 Select this option to instruct Orpheus to check for standard tubing collapse criteria when modeling in conditions of excess annular pressure and axial tension. Output is the collapse factor in the graph window (graph and modeled data) display for running Trip In and Out.

Click the ellipses () associated with any item to see/input additional options for that item's effect on calculations.

5. Select Calculation Resolution; input/select settings to adjust the calculation resolution when modeling complex cases, such as passage of a long toolstring through a small downhole restriction.

 - Maximum RIH-POOH depth increment (CT, JP, WL)

 This depth increment is used as the maximum increment in forming the depth array for tripping calculation.

 - Vary depth increment with well geometry (CT, JP, WL)

 Choose this option to insure that the tripping simulation depths coincide with all constrictions and expansions in the wellbore. This can be a crucial factor in the presence of significant annular flow or high dogleg severity.

 - Allow secondary depth increment (CT, JP, WL)
 - Secondary depth increment
 - Start depth
 - End depth

 A secondary depth increment may be specified for a region of the well. This can provide a greater density of simulation values where needed without significantly increasing total calculation runtime.

- Maximum calculation length (CT, JP, WL)

 Orpheus divides the pipe/cable and the toolstring/BHA into segments to perform forces calculations while the toolstring/BHA is at a specific depth. The length of the segments will not exceed this maximum calculation length.

Advanced Options set the calculation parameters.

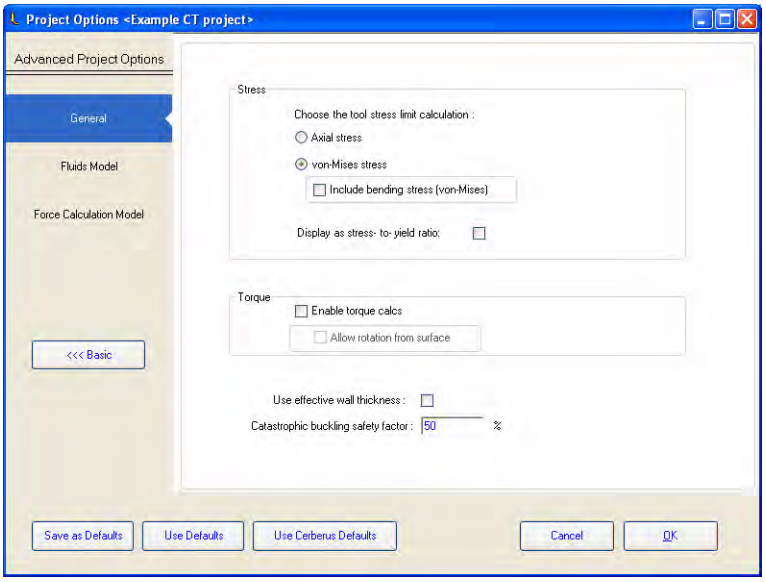
Setting Advanced Options/Calculation Parameters

As mentioned above, Advanced Options set the calculation parameters.

➢ To set advanced options

1. Click the **Advanced** button within the lower left-hand panel of the Basic Project Options view.

 The Advanced Project Options view of the Project Options display screen appears.



*The **Basic** button points back to the Basic Project Options.*

2. From the default Advanced Options tab, input the following:

 - Stress (CT, JP, WL)
 - Axial stress
 - Von-Mises stress
 - Include bending stress (von-Mises)

 Axial stress is caused by the real axial force applied to the pipe or the cable. Von-Mises stress is based on the combination of the three principle stresses (axial stress, radial stress, and hoop stress) and the shear stress (if torque is present). Usually, if wellhead pressure and/or pressure loss (in the pipe or annulus) is high, then von-Mises stress should be selected since in these situations radial stress and hoop stress are high.

 The bending stress mentioned in the option is the additional bending stress caused by the curved section of the wellbore.

 - Tool yield safety factor (WL)
- Enable torque calculations (CT, JP)

 When this option is selected, users can specify torque on end from the Calculate tab.

 - Allow rotation from surface

 When this option is selected, users can specify surface rotation speed from the Calculate tab.
- Use effective wall thickness (CT)

 Effective wall thickness is defined as minimum wall thickness less wall reduction. When enabled, tubing internal diameter is calculated as nominal ID less double of effective wall thickness.
- Catastrophic buckling safety factor (CT)

 This safety factor is used in calculating the catastrophic buckling load, which equals the theoretical buckling load multiplied by the safety factor.

3. Select the Fluids Model tab; input [or alter] information, or click (to check) toggleboxes as needed:
 - Pressure calculations (CT, JP)

 The user can select “static pressure” or “circulating pressure.” When “static pressure” is selected, the fluid is not flowing and no hydraulics calculation is made. When “circulating pressure” is selected, all pressure loss and fluid forces (shear drag, lift force, and form drag) are calculated and are used as input to the tubing forces model calculation.
 - Pipe surface (CT, JP, WL)
 - Cable roughness (WL)
 - Inside pipe roughness (CT, JP)

 Users can select “ignore roughness” or “enter roughness.” When “enter roughness” is selected, the user must input the roughness values for inside the pipe and annulus, respectively. Roughness has no effect on laminar flow but is involved in pressure loss calculation for turbulent flow conditions.

 In this version of the software, pipe roughness is ignored for power law fluids and foam models.
 - Annulus (CT, JP, WL)
 - Eccentricity (CT, JP, WL)

 Eccentricity equals zero for a pipe centered in the well and equals one when it is fully eccentered. This setting only affects the fluid drag calculation. A tool (or cable or pipe) in the center of the well experiences greater fluid drag than when it is lying against the wall. Hence, for worst-case planning purpose, an eccentricity of zero is recommended.
 - Equivalent annular diameter (CT, JP, WL)

 This option is used to select the calculation method used for determining the equivalent pipe diameter. Annular fluid flow calculations are performed by calculating a pipe diameter that would give equivalent pressure-drop values. Equivalent annular diameter is defined so that friction calculation equations for pipe flows can be extended for annular geometry. In general, the equivalent diameter for annular flows can be represented as $De = f(D2, D1)$, where $D2$ and $D1$ are external and internal diameters of the annulus.

 The common approach is to use the hydraulic diameter, where $De = D2 – D1$. However, other definitions of the equivalent diameter exist and several have also been provided in Orpheus.
 - Use correction factor in pressure drop (CT, JP, WL)

- Fluid (uneditable) (CT, JP, WL)
- Correction factor (uneditable) (CT, JP, WL)

This option allows the user to enable or disable the correction factor that is defined for each fluid in Fluid Manager. For ease of use, the correction factor of the currently selected fluid is displayed on-screen. A correction factor of 1 means that the calculated friction pressure will not be modified. A correction factor of 0.6 means that only 60% of the calculated friction pressure drop will be used in the pressure profile.

- Temperature effects on viscosity (CT, JP, WL)

 This option allows the user to enable or disable the effect of temperature on the viscosity of the fluid. For this option to function properly, the fluid must have its temperature properties defined in Fluid Manager. When enabled, the user-entered viscosity profile, instead of constant viscosity, is used in hydraulics calculation.

 The option is only for fluids of liquid type.

- Drag reduction for power law fluid (CT)

 Many polymer fluids, such as guar, xanthan, PHPA, and HEC and surfactant-based fluids exhibit significant drag reduction under turbulent flow conditions. As a result, the traditional Dodge-Metzner correlation overestimates pressure loss of turbulent flow. For these fluids, the effect of drag reduction should be included. This option allows the user to model this effect.

4. Select the Force Calculation Model tab; then click (to toggle or check) toggleboxes as needed:
 - Tool / BHA Model
 - Tool model v1.0 – v1.0 represents the tool as a simple beam element and applies standard equations for beam deflection. For complex tool strings with variable geometry, v1.0 must apply some approximations to average the contact forces due to bending. (CT, JP, WL)
 - Tool model v2.0 – v2.0 is more sophisticated and analytically exact because it applies an FEA-based approach, which accounts for the specific geometry and stiffness of each tool segment and the radial clearances in the wellbore. It is the recommended model for complex tool strings, although it requires more computation time. (CT, JP, WL)
 - Lockup Model (CT, JP)
 - Lockup model v1.0 – Lockup assumed to occur when string yields in compression.
 - Lockup model v2.0 – Lockup assumed to occur when weight transfer from surface to end of string is less than specified amount (recommended).
 - Helical Buckling Load
 - Post-buckled unloading
 - Pre-buckled loading

 There are two ways to define helical buckling load. Chen and Cheatheam (SPE 19176) derived a transition force limit between helical and sinusoidal buckling in an inclined well. However, subsequent research (see SPE 29462) has established that a greater compressive force limit is actually required for sinusoidally buckled tubing to snap into a full helix configuration. Thus, the lower Chen & Cheatham limit can be understood as a post-buckled unloading limit, whereas the higher buckling load is a pre-buckled loading limit.

 Both are realistic in certain circumstances. However, the post-buckled unloading limit tends to be a conservative assumption.

- Stiff String Model – Orpheus includes a stiff string model, which approximates the bending stiffness effects for pipe/tubing in curved well segments. When enabled, Orpheus automatically switches to the stiff string model based on the minimum dogleg severity parameter specified here. (A related bending model is applied to the BHA separately.) (CT, JP)
 - Include stiff string model in forces calculations
 - Minimum dogleg severity for stiff string calcs

5. Select the Wireline tab; input [or alter] information, or click (to check) toggleboxes as needed (WL):
 - Define ‘compression’ at

 Compression, by conventional definition, begins at zero cable tension. However, to minimize the risk of compression occurring, the user may prefer to include an additional safety factor by modeling the job using a higher compression threshold.
 - Weak point safety factor

 The purpose of the weak point safety factor is to avoid premature release of the wireline. It prevents utilization of the full weak point low load limit rating on the wireline when trying to free stuck tools from the wellbore.
 - Maximum allowed speed RIH

 This user-input number provides a realistic limit on the reported speed to avoid nonsensical modeling results.

 The model can calculate the speed at which the fluid drag acting on the tool exceeds the downward pull, causing the tool to move slower than the speed at which cable is being spooled into the well. The result may be bird caging of the cable above the tool. This is more likely to occur in situations where there are tight clearances between the tool and tubing or casing (for example, when running a packer or junk basket). In other situations with greater clearance, there may be no such practical limit to the cable speed, and the calculated speed reported by the model would be unrealistically high.
 - Use effective axial force to calculate stretch

 When this option is enabled, wireline stretch is calculated based on ‘real’ axial force. Theoretically, strain (and therefore stretch) should be calculated using real axial force. Note the word “real” does not mean “field-measured” forces (i.e. weight indicator readings); rather, it means the axial force including the effect of any fluid pressure. The purpose of this option is based on reports from the field, indicating that in some cases stretch data is closer to that calculated using real axial force.
 - Calculate buoyant weight based on weight in water
 - Weight in air (uneditable)
 - Weight in water (uneditable)

 The weight in water value can be used to improve the calculation of wireline buoyant weight. When the option is selected, Orpheus uses the values of weight in air and weight in water to back-calculate an equivalent hydraulic diameter of the wireline. Then the equivalent hydraulic diameter (instead of the nominal diameter) is used to calculate the buoyant weight and axial force.
 - Drum Crush
 - Drum crush threshold
 - Drum crush caution
 - Drum crush warning

Specific wirelines are crush-resistant and should have higher drum crush values than standard wireline. Drum crush may occur if cable being POOH at high tension is spooled onto the drum on top of cable that was spooled at a much lower tension. Orpheus predicts RIH and POOH tensions and can warn the user when one value is much larger than the other.

Threshold: minimum tension percentage of cable breaking strength at which cable damage due to crushing may occur.

Caution: occurs when threshold is exceeded and POOH tension is this multiple greater than RIH tension.

Warning: occurs when threshold is exceeded and POOH tension is this multiple greater than RIH tension.

Setting Project-option Defaults

1. Select the **Save as Defaults** button in any preference-group dialog to retain user-defined properties as defaults (properties that are used for all new projects in Orpheus).

2. Upon re-accessing the preference-group dialog for future projects, select **Use Defaults** to populate related fields with previously saved default properties.

 OR

 Select **Use Cerberus Defaults** to populate all the fields within any dialog with Cerberus default-property information.



The buttons described above pertain to all advanced options; not just the options visible to the user.

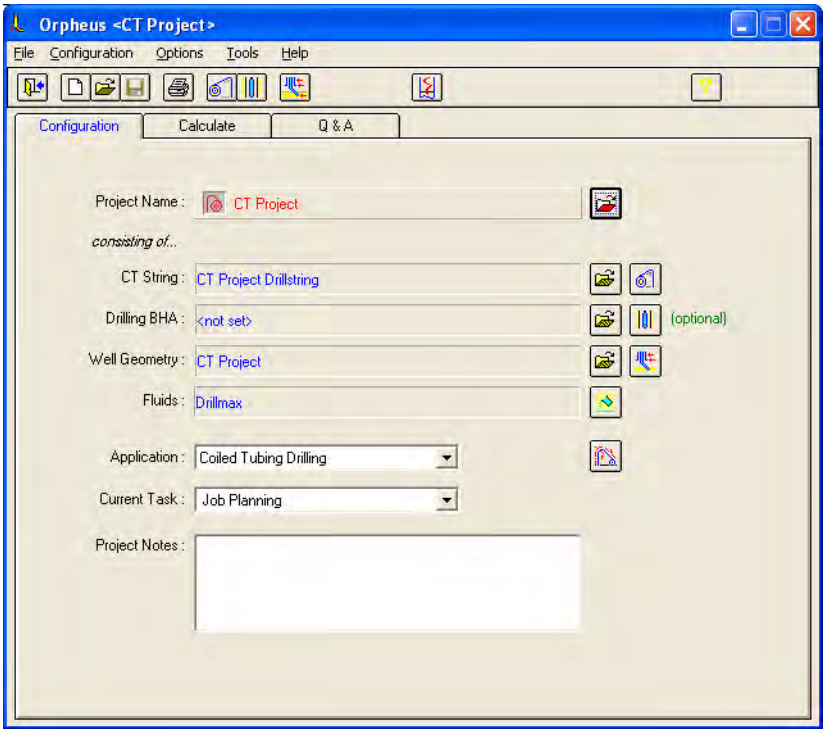


Hypothetical fluid-hydraulics situations may be modeled/calculated in Orpheus to determine the effects of differing variables. Please see the appendix Advanced Fluids-model Options in Orpheus for additional information.

Performing Calculations

Various methods are available in Orpheus to perform calculations. These methods are accessible through an Orpheus project display screen similar to the following:

Orpheus Project Display Screen



Calculation Methods

The following calculation methods are available:

- Standard Calculations, accessed via the **Calculate** tab.
- Analysis Tools, accessed via the **Calculate** tab > **Calculation Tools**

 OR via the Orpheus Tools menu-bar item.
- User-friendly question-and-answer wizards, accessed via the **Q & A** tab.
- Reports, accessed via the menu bar or the tool bar (see page 131).

Calculations are not stored with a project; they must be recalculated for future use.

If a CT String is of type FLATpak™, hydraulics and catastrophic buckling calculations are not available with this release of Cerberus.

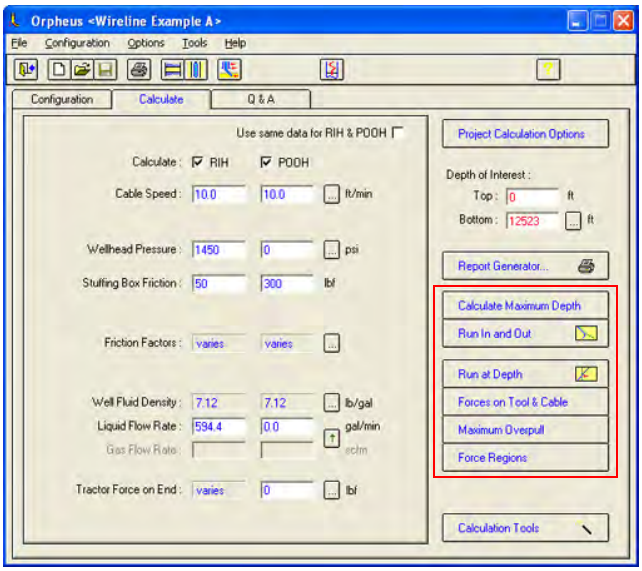
Standard Calculations

To gain access to standard calculations, select the **Calculate** tab from an Orpheus-project display screen. Standard calculations are accessed via buttons along the right-hand portion of the Calculate-tab display

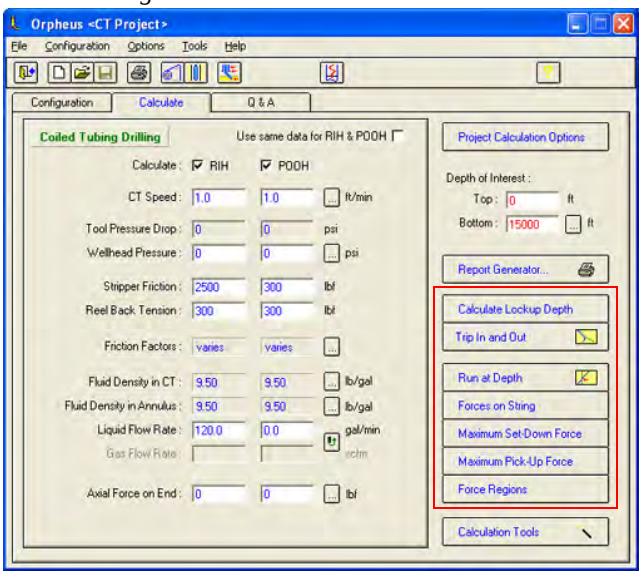
screen. The only exceptions are the **Report Generator** and **Calculation Tools** buttons, which access other distinct calculation methods.

Slight differences exist among the conveyance methods' standard-calculation display-screen appearances and procedures:

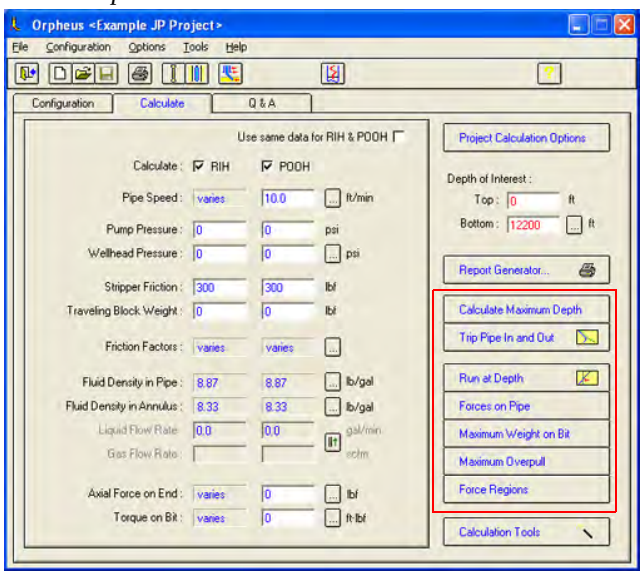
Wireline Standard-calculation Buttons



Coiled Tubing Standard-calculation Buttons



Jointed Pipe Standard-calculation Buttons



Prior to using the functions available in either the **Calculate** tab or the **Q&A** tab, the user may input (from the right-hand side of the Calculate-tab panel) **Top** and/or **Bottom Depths of Interest** to affect calculations of type “tripping” or “running.” “At depth” calculation results do not reflect **Depth of Interest** designations.

These designations affect tripping or running calculations viewed via the **Report Generator** button, and several tripping or running functions in the **Calculation Tools**:

- Sensitivity Analysis
- Report Generator
- Weak Point Selection
- Pump Down Rate

After a button is clicked, note the availability of tools to assist the user’s customization of the graph within the display screen.

Calculate Lockup Depth (CT)/Calculate Maximum Depth (WL/JP)

The lockup depth calculation shows the depth at which the CT locks up. When the tool string reaches this depth, it goes no deeper, even though CT may still be entering the well at the surface.

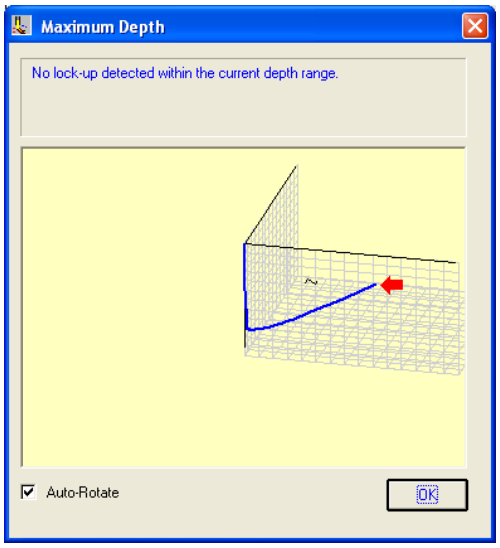
The maximum depth calculation shows the depth at which the wireline either enters compression or exceeds the tensile limit.

➢ To calculate lockup depth/maximum depth

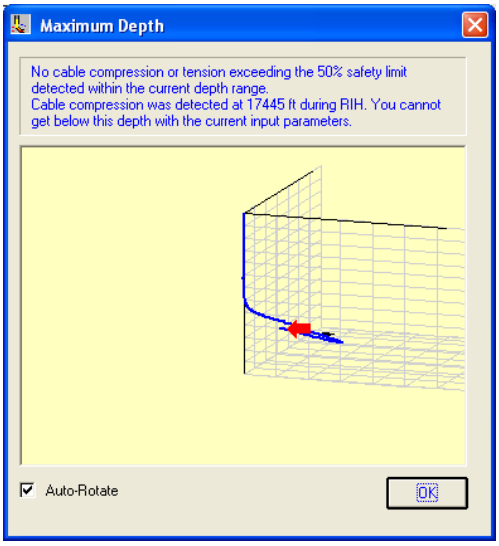
Select **Calculate** > **Calculate Lockup Depth** (CT) or **Calculate Maximum Depth** (WL/JP).

*On the **Q & A** tab, select **Getting into and out of the well**. Then select "**Can I reach TD...?**”*

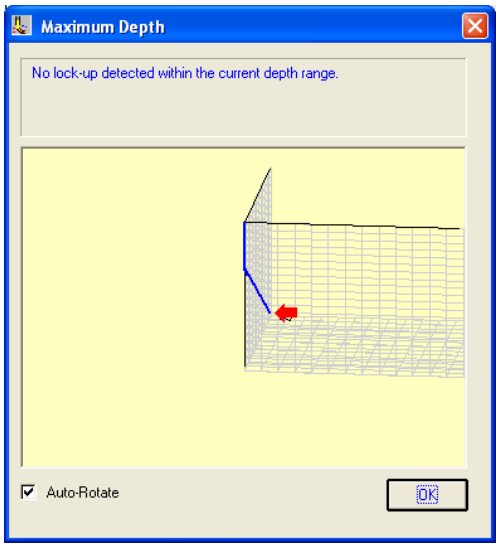
Coiled Tubing Lockup Depth



Wireline Maximum Depth



Jointed Pipe Maximum Depth



Trip In and Out (CT and JP)/Run in and Out (WL)

The Trip/Run In and Out calculation provides graphs of surface weight plotted against depth as the tools are tripped/run into and out of the well. The calculation is performed from the surface to the measured depth, providing a quick overview of the surface weights to expect for all depths of the job.

If lockup is detected (CT) at a depth shallower than the measured depth, a warning message is displayed and the measured depth is reset to a value slightly lower than the lockup depth before the application resumes calculations.

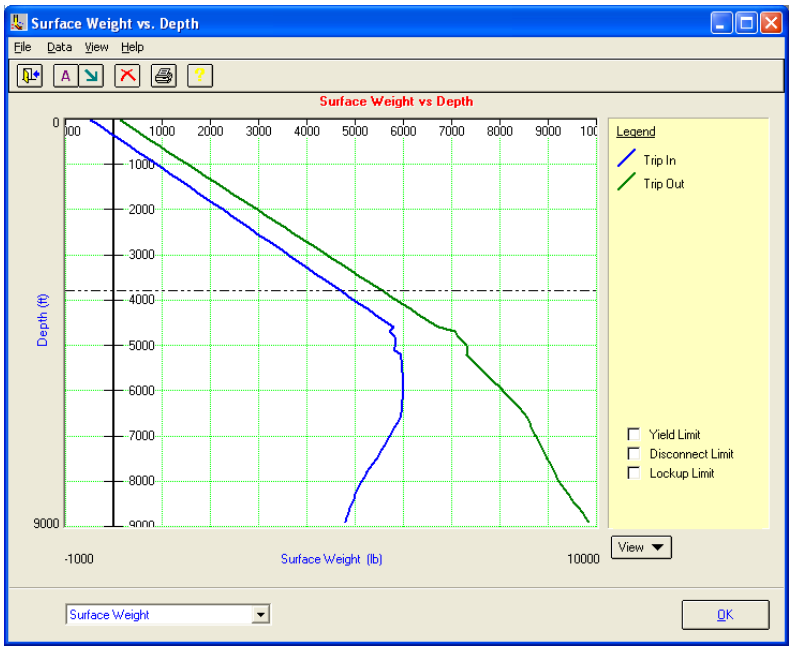
If the maximum depth is detected (WL) at a depth shallower than the measured depth, the graph indicates the maximum depth. The curve then continues in gray to the measured depth, as if the wireline could withstand compression.

➢ To calculate trip/run in and out

Select the **Trip/Run In and Out** button.

*On the **Q & A** tab, select **Getting into and out of the well**. Then select one of the "**Plot a graph**" scenarios.*

Surface Weight vs. Depth (CT)



Surface Weight vs. Depth (WL)



The Surface Weight graph plots weight against depth as the CT/WL is tripped/run into the well (shown in blue) and out of the well (shown in green). The calculation is performed from the surface to the measured depth.

Orpheus can calculate the lower compression and upper yield limits on the surface weight as the CT/JP/WL is tripped/run in and out of the well. By default, Orpheus does not calculate these limits because they dramatically increase the calculation time. NOV CTES recommends calculating them only when needed.

Orpheus does not consider the capabilities of the surface equipment; it only considers the properties of the conveyance and tool.

Field Data Input

For the Surface Weight vs. Depth calculation (Trip In and Out (CT and JP)/Run in and Out (WL), data can be created from scratch or imported from an external file (ASCII text or Orion database).

➢ To access the field data-entry area

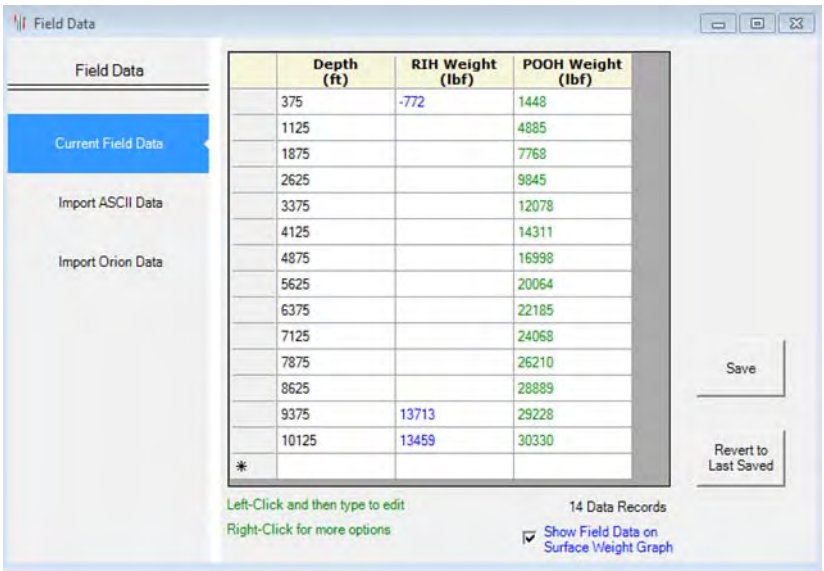
1. Select the Calculate tab within an Orpheus project.
2. Click **Trip In and Out** (CT and JP)/**Run in and Out** (WL).
3. Select **Data > Field Data** from the menu bar.

Manual Field Data Entry

➢ To manually enter data

1. Click any cell to modify the data in that cell.
2. Right-click the left-most column to access a menu of options for that row. The row can be deleted, moved up, or moved down. A blank row can be inserted above or below an existing row.
3. **Save** the data at any time or **Revert to Last Saved** data to undo the latest changes.
4. Click (to check) **Show Field Data on Surface Weight Graph**, which shows data as described when exiting.

Current Field Data



Import ASCII Text Data

➢ To import ASCII text data

1. Click **Browse** to select a file name.

2. Use the associated dropdowns to select **Column Number** and **Units** for the **Depth** and **Weight** fields. When importing ASCII text data, the file can contain any number of fields in any order as long as it contains the **Depth** and **Weight**. Only the columns for the **Depth** and **Weight** need be specified, since all other data columns are ignored.

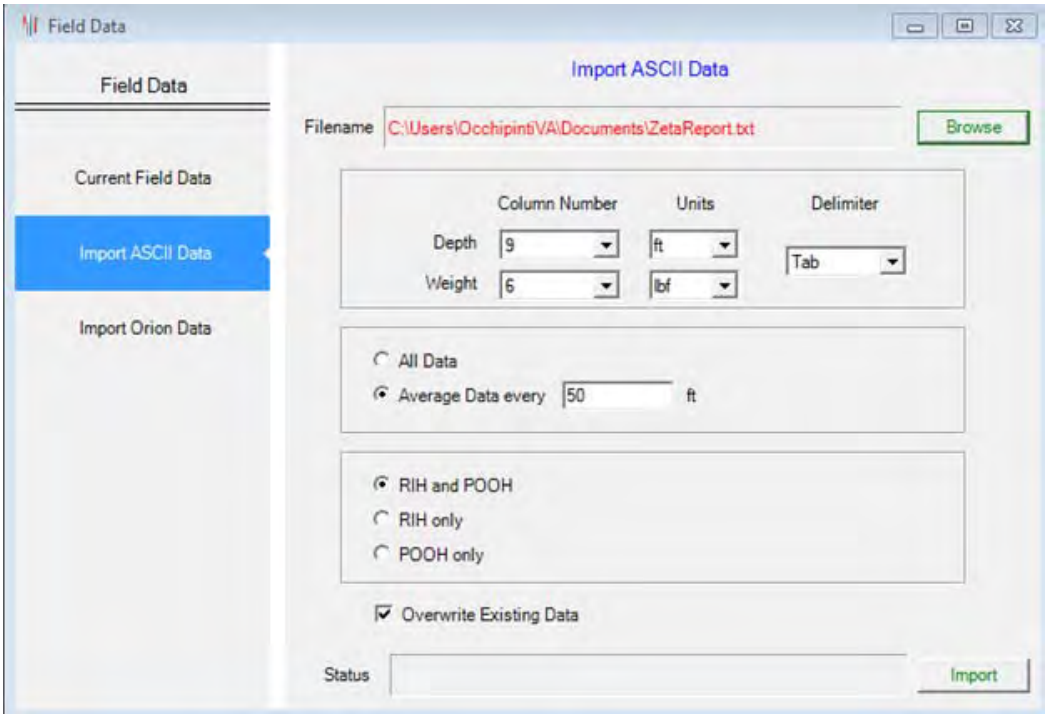
 a. The **Units** of data are specified separately and can be either Metric or English. The units do not have to be the same and do not have to match the Orpheus units setting. If they are different from the Orpheus units, they are converted to Orpheus units.

3. Set the **Delimiter** between data columns using the associated dropdown.

4. Click (to toggle) **RIH and POOH**, **RIH only**, or **POOH only** to specify the weight calculations desired.

5. Click (to check) **Overwrite Existing Data** if so desired. To append data to the existing data, do not check the “overwrite” checkbox. When data is appended, multiple files may be imported and merged together.

Import ASCII Data



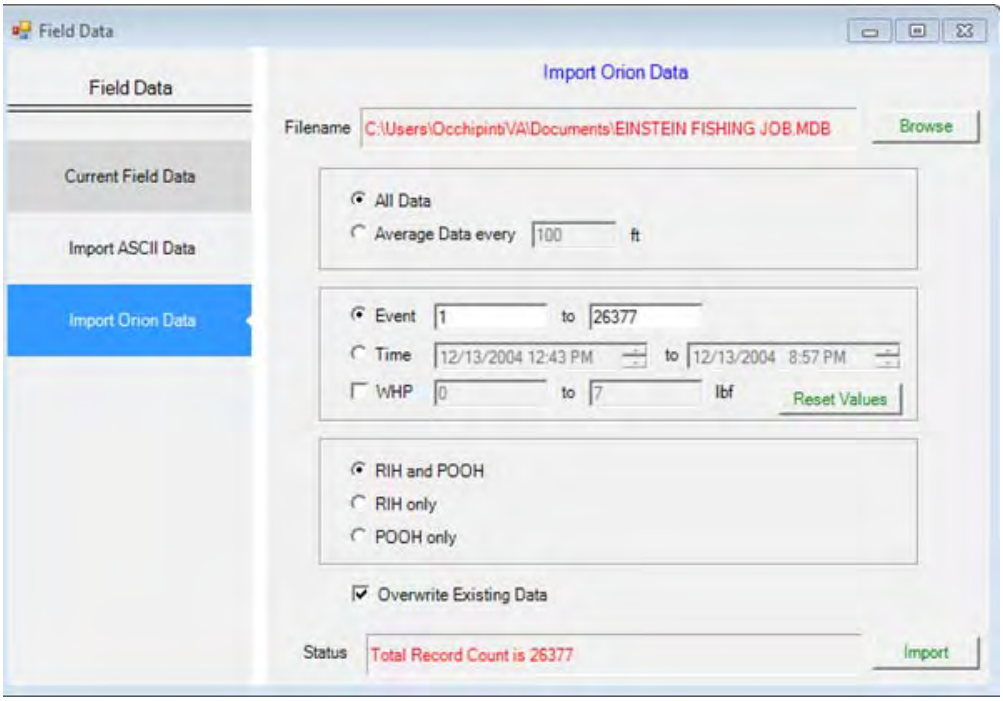
Import Orion Database

Import data from an external Orion/OrionNET file, either in the old MDB format or in the newer ONET format. Options to process all data or to average the data across a certain depth range are available.

➢ To import Orion data

1. Click **Browse** to select a file name.
2. Select filtering requirements for the data.
 a. Input **Event Min** and **Max** (MDB only).
 b. Click (to toggle) **Time** and select a range.
 c. **WHP** (if WHP exists) range is displayed and may be altered; click **Reset Values** to reset if altered in error.
 d. **Weight** can be calculated for **RIH**, **POOH**, or both.
 e. Click (to check) **Overwrite Existing Data** if desired. If data is not overwritten, it is appended to the current data, thus allowing for multiple files to be imported and merged together.
3. Click **Import**.

Import Orion Data



Setting Graph Configurations



See "Setting Graph Display Options in Orpheus" on page 379 for information concerning altering the way calculation results are displayed graphically.

From the graphical representations, the user may set graph configurations to plot the scale or to zoom out of a zoomed graph.

➢ To set graph configurations

1. Right-click on a graph.
2. Select either **Scale** or **Zoom Out**.
 a. If **Scale** is selected, a Plot Scale popup appears, from which the user may specify **Minimum** and **Maximum Tension** and **Minimum** and **Maximum Depth**.
 b. Selecting **Zoom Out** is effective only if the graph has been zoomed using the mouse click-and-drag method on the graph surface.

Run at Depth

The Run at Depth calculation in Orpheus provides a look at the forces, stresses, and other parameters along the length of the string while the string is positioned at a given depth. This calculation uses the user-specified force on the end of the string. After the calculation is performed, the user may modify the depth, force on end, or surface weight from the output graph screen. This allows the user to enter a known surface weight and to see the expected force on the end of the string – particularly useful in stuck-string situations where the user wants to know what tension the string is in while pulling a specific value at surface.

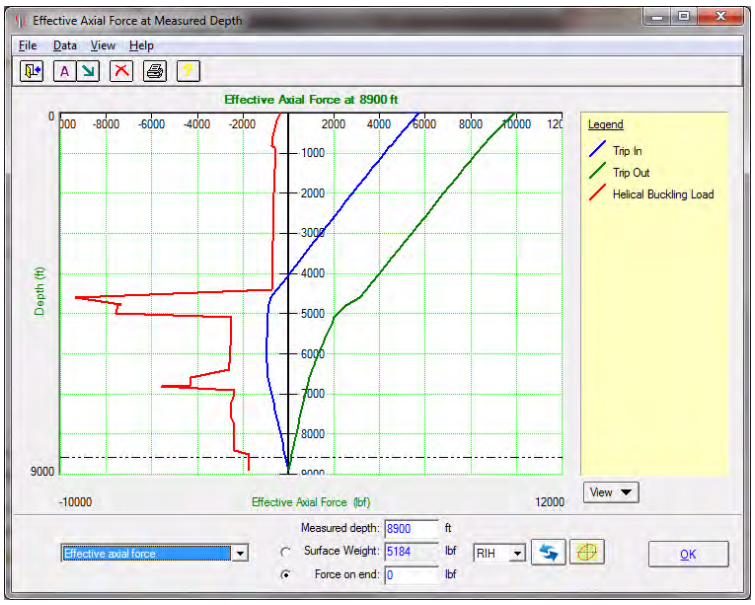
➢ To calculate Run at Depth

1. Select the **Run at Depth** button to observe, in graphic format, the Effective Axial Force at Measured Depth (default) display screen.



*On the **Q & A** tab, select **Performing the job at depth**. Then, click **Plot a graph of CT Tension vs. Depth** (CT), or **Plot a graph of Cable Tension vs. Depth** (CT).*

Coiled Tubing Run at Depth Graph



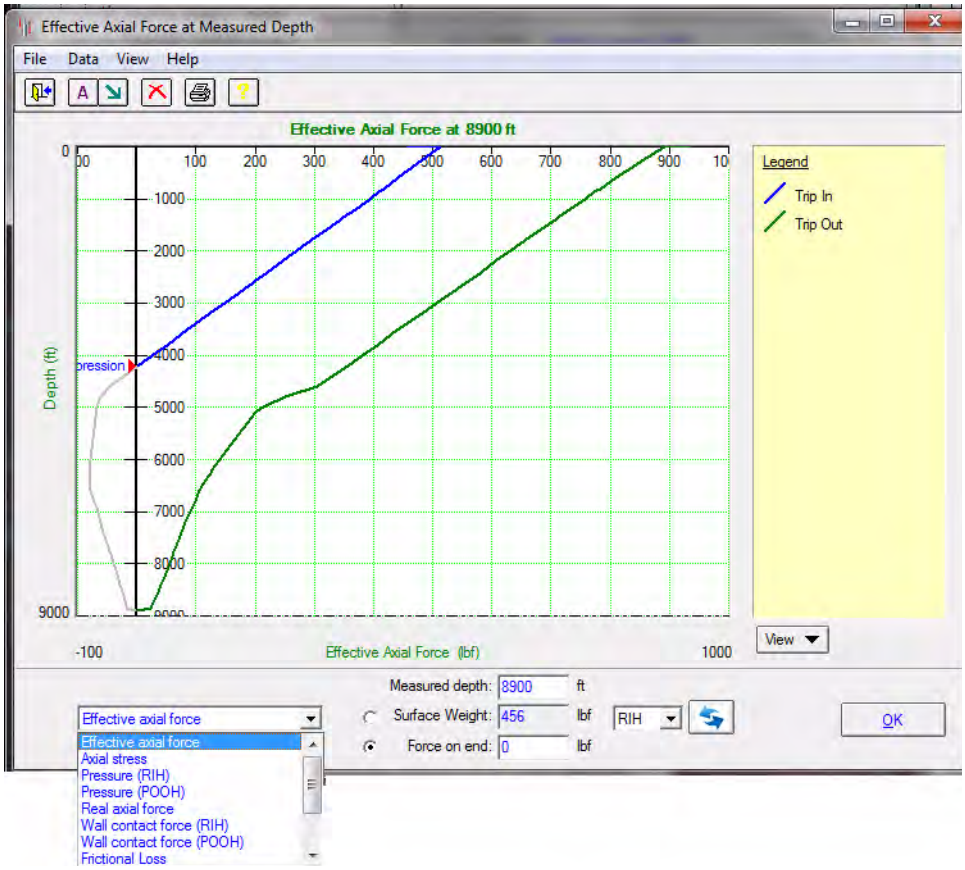
In the CT graph, if the Helical Buckling Load (red line) is greater than the axial load while RIH (blue line), then helical buckling is occurring at that section of the tubing, although lockup has not necessarily occurred.

"Effective" describes the actual production of, or the power to, produce an effect. The effective axial force takes into consideration all of the calculations/graphs available from the dropdown to arrive at the closest calculation of axial force possible.

For more information concerning the effective force, see the NOV CTES "tech note" *The Effective Force* by Ken Newman and Kenneth Bhalla, dated January 13, 1999:

http://ctes.nov.com/Documentation/technotes/Tech%20Note%20Effective%20Force.pdf

Wireline Run at Depth Graph



2. The user may view the default **Effective axial force** graph, or may select another type of chart (from the dropdown) to view calculated forces exerted the length of a conveyance at a specified measured depth.

 a. Within the default **Effective axial force** graph, the user may:

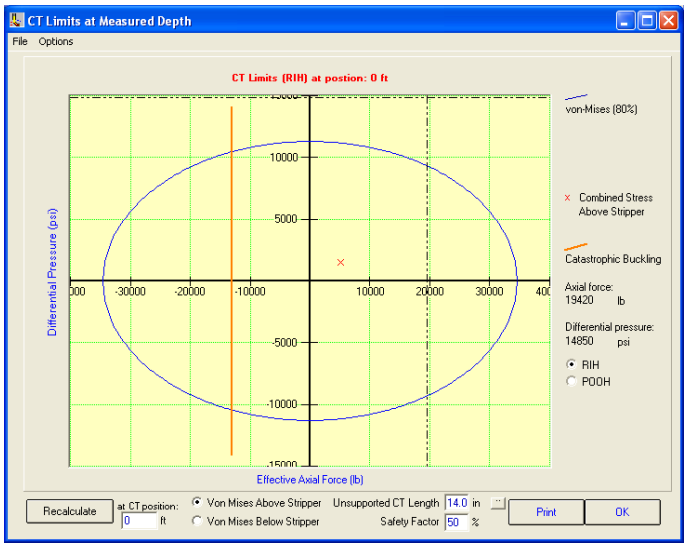
 - Change Measured depth; then click the **Calculate and Refresh** icon ().

The measured depth default value is the maximum reachable depth within the depth range.

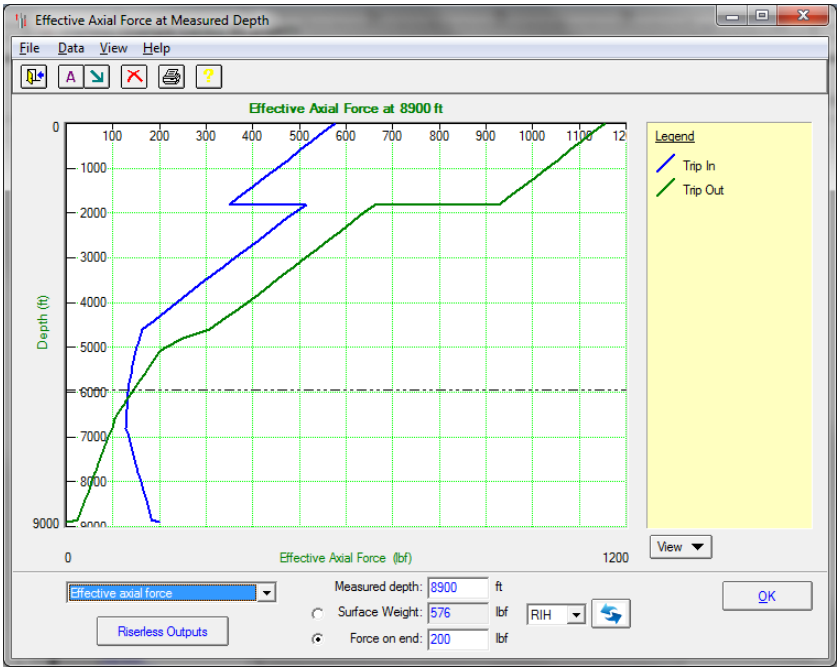
 - Click (to toggle) the **Surface Weight** field and enter the desired weight.
 - When the surface weight is experienced while attempting to pull out of hole, select **POOH** from the dropdown; then click the **Calculate and Refresh** icon ().
 - Click (to toggle) the **Force on end** field and enter the desired force.
 - When the **Force on end** is experienced while attempting to pull out of hole, select **POOH** from the dropdown; then click the **Calculate and Refresh** icon ()

When the Force on end becomes too negative in CT or JP mode, the string locks up. In this situation, the program calculates an approximate Surface Weight, overriding any user surface-weight input. The user can verify the values displayed on the screen by viewing the top and bottom effective force from the modeled data grid.

3. (Optional: CT /JP) Click the **CT Limits/Pipe Limits** icon () to view limits at measured depth.

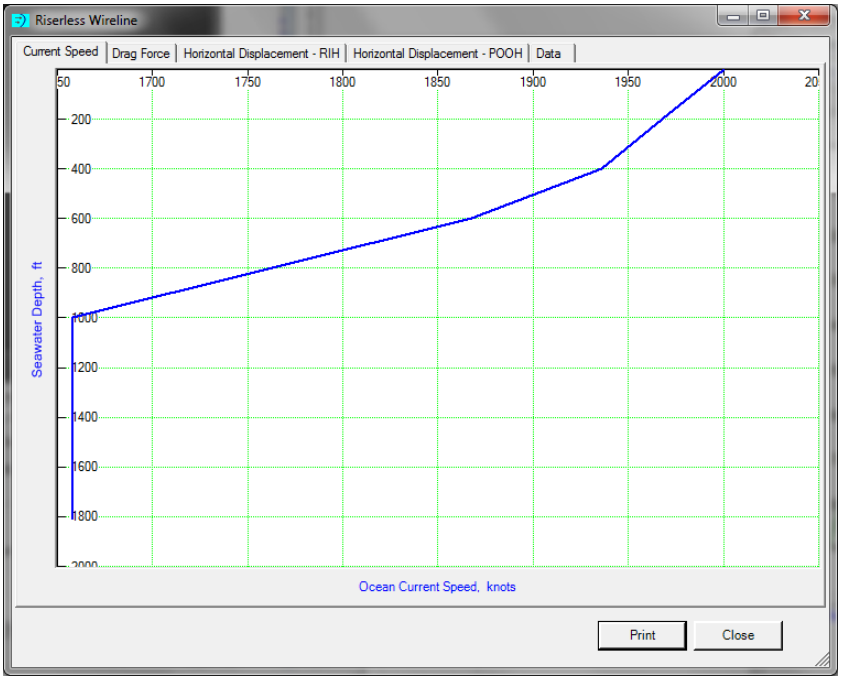


4. (Optional: WL) If the well is **Riserless**, the Run at Depth calculation results appear as follows:



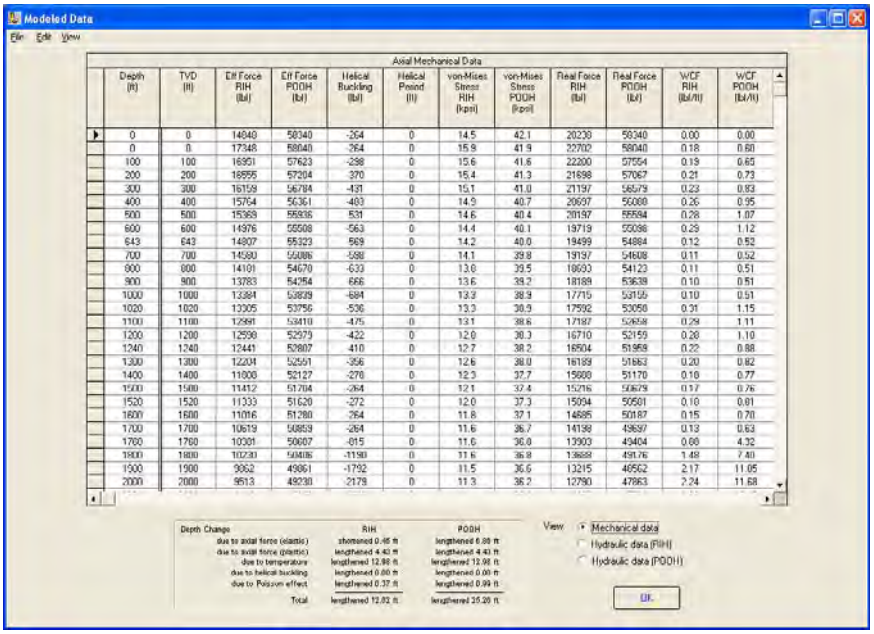
The discontinuity is attributable to the pressure difference at the wellhead.

a. Select the Riserless Outputs button, and then click each tab, to see the various available graphs.



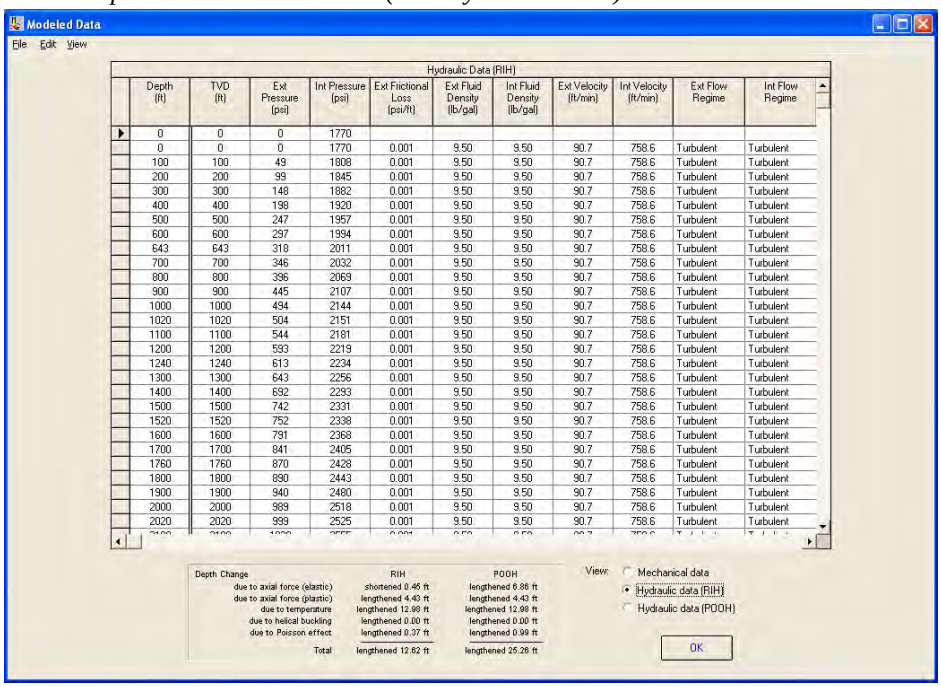
5. View the modeled data table(s) by clicking, within the menu bar, **Data > Modeled Data**.

Run at Depth: Modeled Data Table (Mechanical Data)



If a Hydraulics-enabled access key is in use (see Fluids Model: Advanced Hydraulics Options), three tables become available: Hydraulics data (RIH), and Hydraulics data (POOH), and Mechanical data.

Run at Depth: Modeled Data Table (RIH Hydraulic Data)



If hydraulics are not enabled, then the **Int** and **Ext** pressure columns appear in the Mechanical data table.

If a CT String is of type FLATpak™, hydraulics and catastrophic buckling calculations are not available with this release of Cerberus.

6. From the graphical representations, the user may set graph configurations to plot the scale or to zoom out of a zoomed graph.

 ➢ To set graph configurations

 a. Right-click within a graph.

 b. Select either **Scale** or **Zoom Out**.

 1) If **Scale** is selected, a Plot Scale popup appears, from which the user may specify **Minimum** and **Maximum Tension** and **Minimum** and **Maximum Depth**.

 2) Selecting **Zoom Out** is effective only if the graph has been zoomed using the mouse click-and-drag method on the graph surface.

Forces on String/Tool and Cable

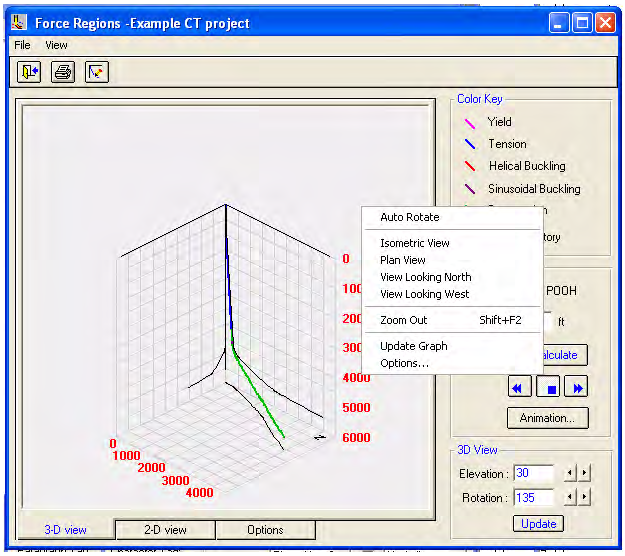
Forces on String/Tool and Cable calculates the regions of yield, tension, compression, helical buckling (CT only), and sinusoidal bucking (CT only), along the length of the conveyance at a specific measured depth.

➢ To calculate forces

1. Select the **Forces on String/Tool & Cable** button.

*On the **Q & A** tab, select **Performing the job at depth**. Then click **View a 3D animation of CT Tension vs. Depth** (CT), or **View a 3D animation of Cable Tension vs. Depth** (WL).*

2. Right-click the graph to adjust the view.

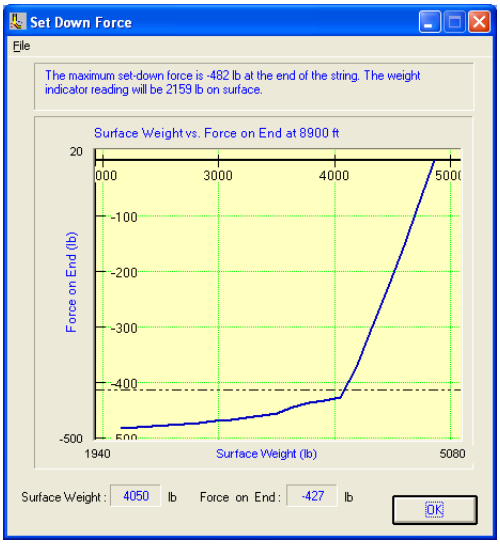


Maximum Set-down Force (CT)/Maximum Weight on Bit (JP)

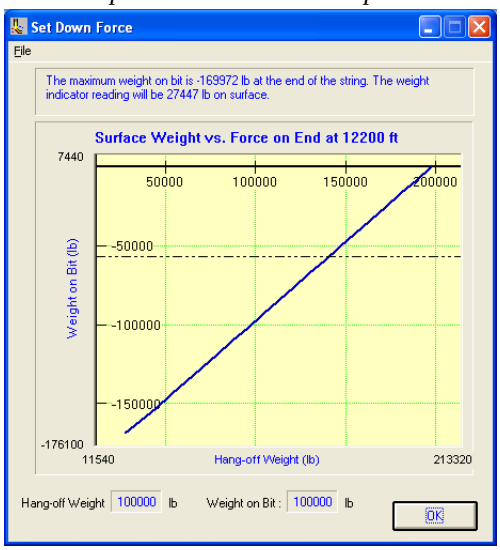
The maximum set-down force is the maximum compressive force (negative) that can be transmitted to the downhole end of the tool string at the measured depth. This downhole weight corresponds to the minimum surface weight at lockup (CT)/lockup limit (JP) on the Surface Weight graph.

Orpheus considers the forces along the entire conveyance, not just at the surface, when performing this calculation.

Coiled Tubing Set Down Force Graph



Jointed Pipe Set Down Force Graph



➢ To calculate set-down force

Select the **Maximum Set-down Force** (CT)/**Maximum Weight on Bit** button (JP).

*On the **Q & A** tab and select **Performing the job at depth**. Then click **How does the set-down force (WOB) vary with surface weight?***

Maximum Pick-up Force (CT)/Maximum Overpull (JP/WL)

The maximum pick-up force is the maximum pull that can be applied at the end of the tool string at the specified measured depth before yield occurs at some point in the conveyance medium.

Orpheus considers the forces along the entire conveyance, not just at the surface, when performing this calculation.

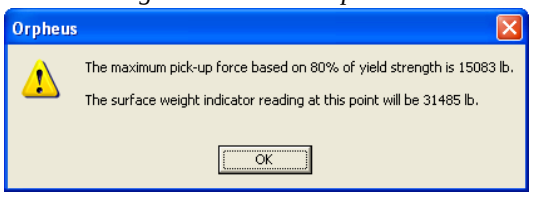
For coiled tubing, ensure that the predicted surface weight does not exceed the maximum pulling force of the injector, although this is rare.

➢ To calculate maximum pick-up force (CT)/overpull (WL/JP)

Select the **Maximum Pick-up Force** (CT)/**Maximum Overpull** button (WL/JP).

*On the **Q & A** tab and select **Performing the job at depth**. Then click **Plot a graph of Maximum Pick-up Force against Depth***

Coiled Tubing Maximum Pick-up Force Calculation



Forces on String (CT)/Tool & Cable (WL)/Pipe (JP)

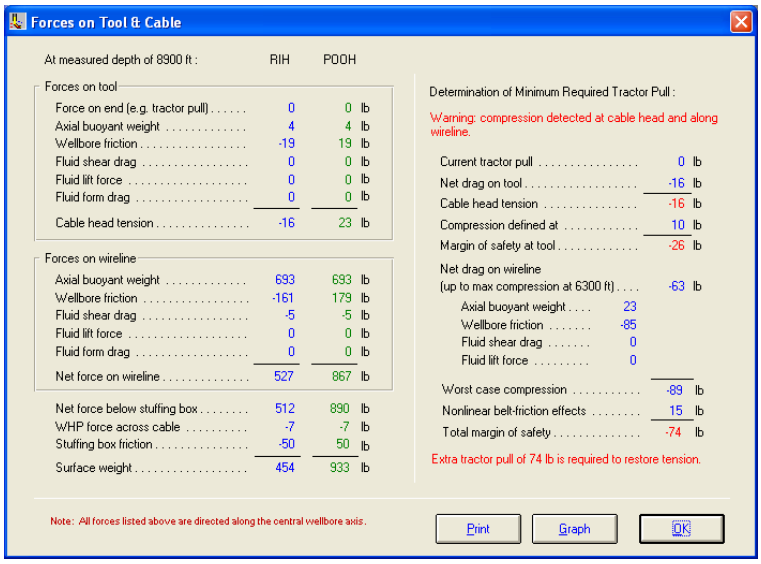
The Forces on String/Tool & Cable/Pipe calculates the net force on the conveyance. The calculation is a combination of tractor pull, buoyant tool and cable (CT) weight, wellbore friction, fluid drag on the tool, and any excess drag on the conveyance (excess drag on the conveyance is any drag over and above the conveyance's buoyant weight).

➢ To calculate the forces on string (CT)/tool & cable (WL)/pipe (JP)

Click the **Calculate** tab, then the **Forces on String/Tool & Cable/Pipe** button.

*On the **Q & A** tab, select **Performing the job at depth**. Then, click **What are the forces acting at this depth?***

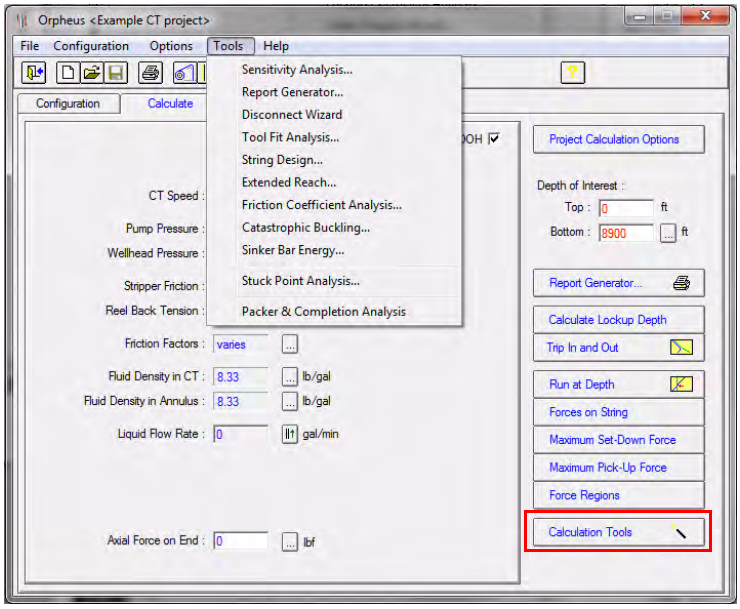
Wireline Forces on Tool & Cable Calculation



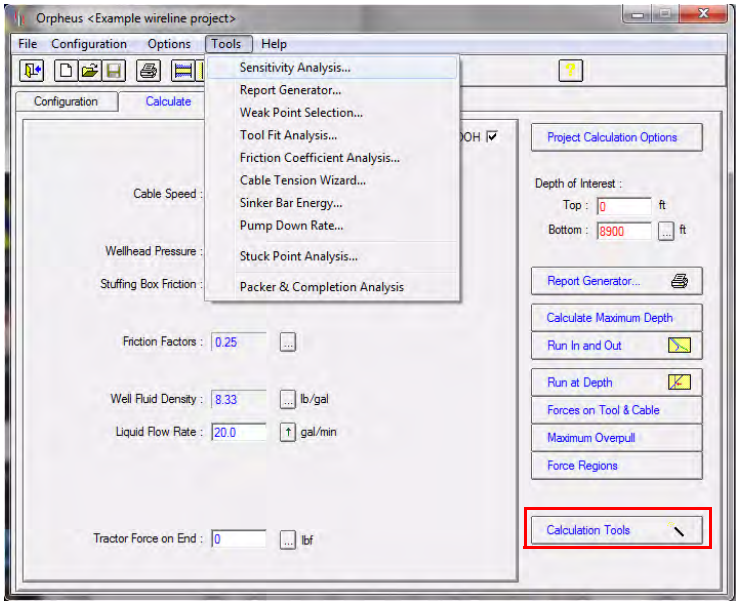
Analysis Tools

Analysis tools are accessed via Orpheus's **Calculate tab > Calculation Tools,** or via the **Tools** menu-bar item.

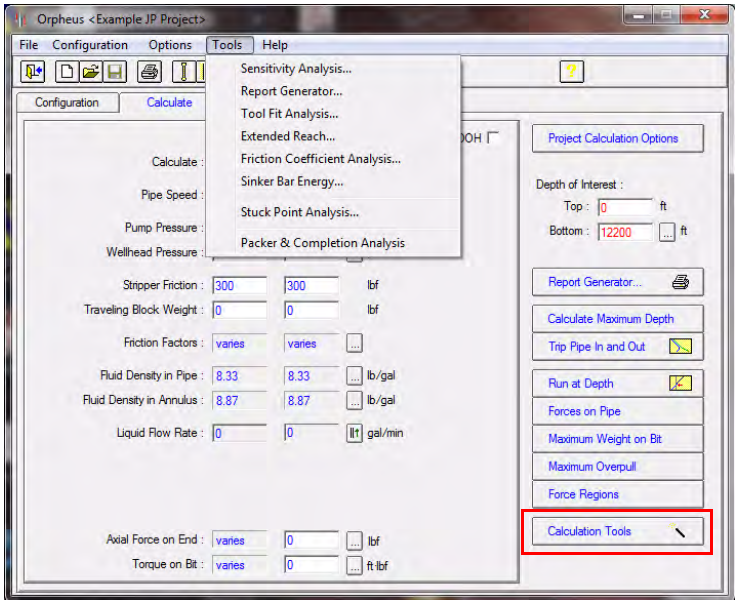
Coiled Tubing Analysis Tools



Wireline Analysis Tools



Jointed Pipe Analysis Tools



Sensitivity Analysis

Use the Sensitivity Analysis Wizard to vary key parameters with the intent of testing their effect on lockup depth (CT), maximum attainable depth (WL), set-down force, pick-up force, and maximum over-pull at the cable head (WL).

Within the interface, three areas or “panels” (Project Information, Sensitivity Setting, and Sensitivity Analysis) reside in one wizard display.

All the areas can be resized, hidden, minimized, or maximized by the user.

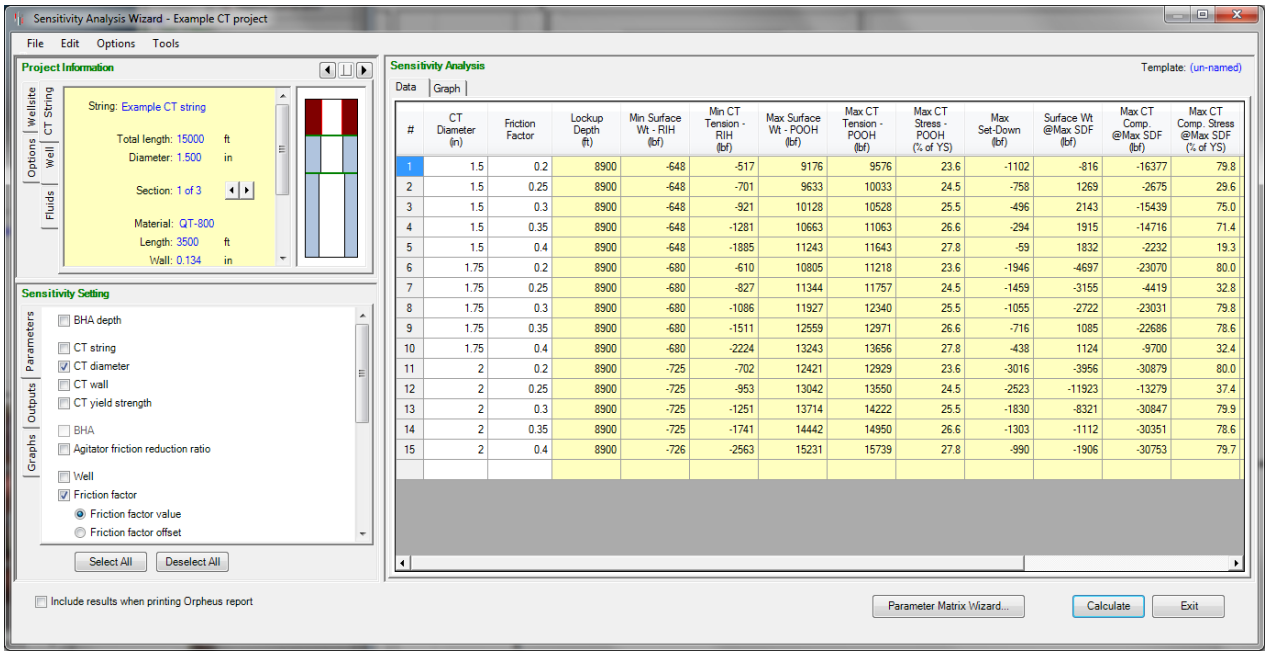
The user may view project information without leaving the wizard. Available project information includes CT/JP string (WL toolstring), BHA, well geometry and survey, fluids, wellsite parameters, and project options. The plots of the CT/JP string, BHA or toolstring, and well geometry and survey are displayed also.

Sensitivity Analysis Wizard: User Interface and Functionality

The wizard window consists of:

1. Top menu-bar
2. Project Information panel
3. Sensitivity Setting panel
4. Sensitivity Analysis panel, including data and graph
5. Bottom panel for main options and command buttons

Sensitivity Analysis Wizard



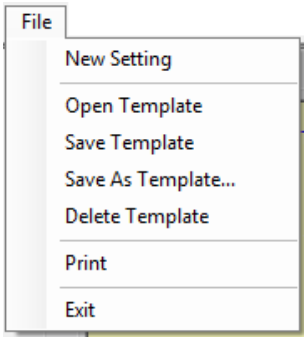
#	CT Diameter (in)	Friction Factor	Lockup Depth (ft)	Min Surface Wt - RIH (lbf)	Min CT Tension - RIH (lbf)	Max Surface Wt - POOH (lbf)	Max CT Tension - POOH (lbf)	Max CT Stress - POOH (% of YS)	Max Set-Down (lbf)	Surface Wt @Max SDF (lbf)	Max CT Comp. @Max SDF (lbf)	Max CT Comp. Stress @Max SDF (% of YS)
1	1.5	0.2	8900	-648	-517	9176	9576	23.6	-1102	-816	-16377	79.8
2	1.5	0.25	8900	-648	-701	9633	10033	24.5	-758	1269	-2675	29.6
3	1.5	0.3	8900	-648	-921	10128	10528	25.5	-496	2143	-15439	75.0
4	1.5	0.35	8900	-648	-1281	10663	11063	26.6	-294	1915	-14716	71.4
5	1.5	0.4	8900	-648	-1885	11243	11643	27.8	-59	1832	-2232	19.3
6	1.75	0.2	8900	-680	-610	10805	11218	23.6	-1946	-4697	-23070	80.0
7	1.75	0.25	8900	-680	-827	11344	11757	24.5	-1459	-3155	-4419	32.8
8	1.75	0.3	8900	-680	-1086	11927	12340	25.5	-1055	-2722	-23031	79.8
9	1.75	0.35	8900	-680	-1511	12559	12971	26.6	-716	1085	-22686	78.6
10	1.75	0.4	8900	-680	-2224	13243	13656	27.8	-438	1124	-9700	32.4
11	2	0.2	8900	-725	-702	12421	12929	23.6	-3016	-3956	-30879	80.0
12	2	0.25	8900	-725	-953	13042	13550	24.5	-2523	-11923	-13279	37.4
13	2	0.3	8900	-725	-1251	13714	14222	25.5	-1830	-8321	-30847	79.9
14	2	0.35	8900	-725	-1741	14442	14950	26.6	-1303	-1112	-30351	78.6
15	2	0.4	8900	-726	-2563	15231	15739	27.8	-990	-1906	-30753	79.7

Menu Bar

➢ To use the Sensitivity Analysis menu bar

1. The **File** menu allows the creation of a new setting, provides various template operations, and provides options for printing the analyses and exiting the application.

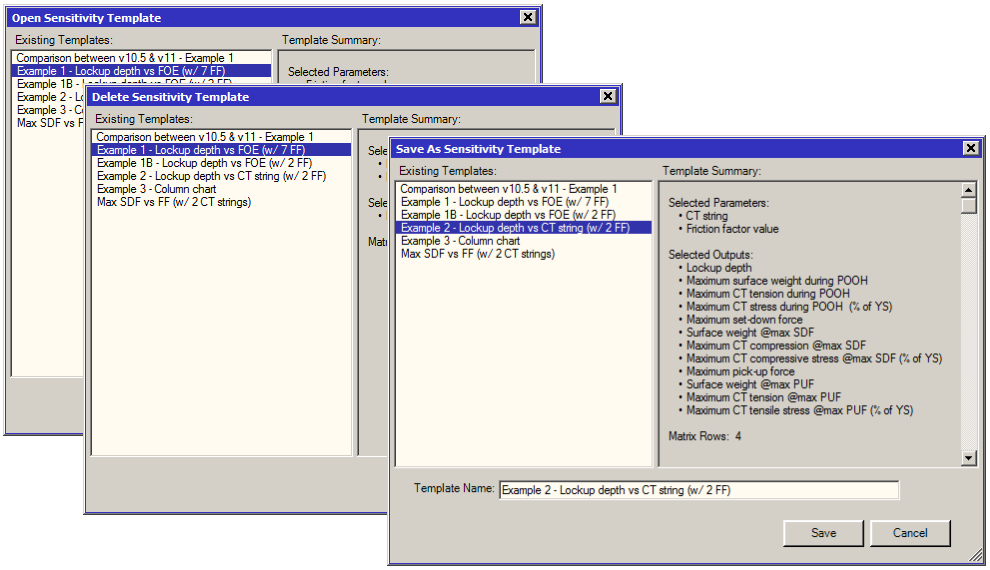
Sensitivity Analysis File Menu



 a. Select **New Setting** to clear out the sensitivity table shown in the Sensitivity Analysis panel of the display. In a cleared-out sensitivity table, **Friction factor** is selected by default as the only parame-

ter and **Lockup depth** is selected as the only output (See “Sensitivity Analysis Panel” on page 109 for information concerning this panel).

b. Select one of the template options: **Open**, **Save**, **Save As**, or **Delete**. In each instance, the Template Summary is shown in the right-hand portion of the dialog.

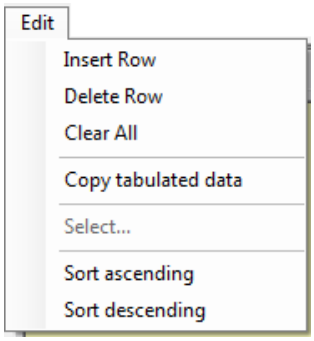


The **File > Save As** option allows the user to input a **Template Name** in the related field.

c. Select **Print** to access the Orpheus Print Manager and to determine print options.

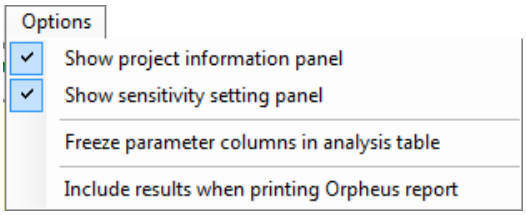
2. The **Edit** menu allows the user to insert, delete, or clear sensitivity table rows; to copy the whole sensitivity table contents (including the column headers) to the clipboard for potential spreadsheet paste; or to sort columns.

Sensitivity Analysis Edit Menu



3. The **Options** menu allows the user to show/hide the project information panel, to show/hide the sensitivity setting panel, to freeze the parameter columns as the user moves toward the right portion of the analysis-table display, or to include the results when printing an Orpheus report.

Sensitivity Analysis Options Menu

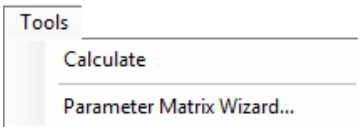


If the last option above (**Include results when printing Orpheus report**) is selected before the user returns to the main Orpheus display, and if the main report is printed from the report generator, then the sensitivity analysis results are included in the printed report.

This option is accessible from both the menu-bar **Options** item and from the bottom panel of the Sensitivity Analysis Wizard.

4. The **Tools** menu allows the user to **Calculate** data or to launch the Sensitivity **Parameter Matrix Wizard**.

Sensitivity Analysis Tools Menu



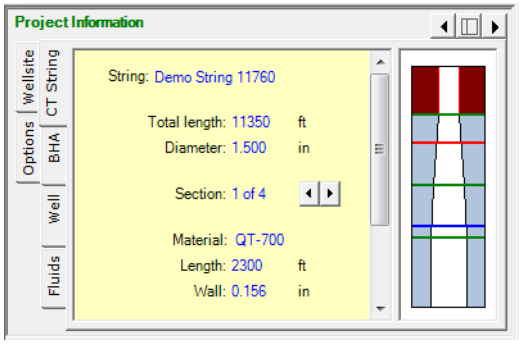
Both of these tool functions are also available via buttons in the bottom panel of the main Sensitivity Analysis Wizard display.

Project Information Panel

Un-editable project information for the user's review is provided in the Project Information panel of the Orpheus Sensitivity Analysis by way of various tabs in the left-hand portion of the panel.

If the project does not contain a BHA (CT)/Toolstring (WL), then no BHA/Toolstring tab appears.

Sensitivity Analysis Project Information Panel

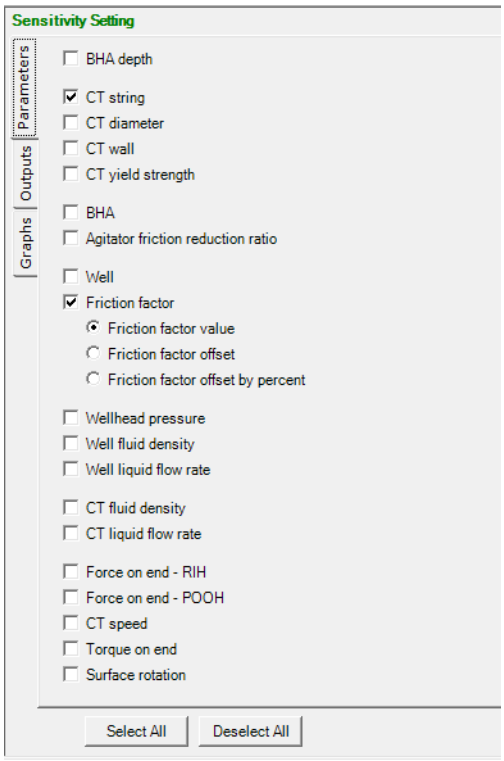


Sensitivity Setting Panel

The Sensitivity Setting panel contains tabs that allow the user to select parameters and outputs for the sensitivity calculation. The **Graphs** tab in the Sensitivity Analysis panel is available only if the number of parameters selected is one or two. (See the Sensitivity Analysis Wizard figure to better understand the wizard layout.)

Parameters selected (checked) within the Sensitivity Setting **Parameters** tab are shown in the Sensitivity Analysis panel's **Data** tab as column headings.

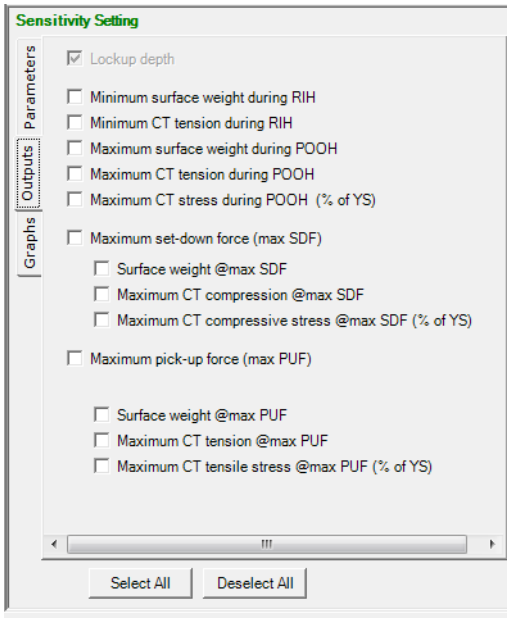
Sensitivity Setting Panel – Parameters Tab



Outputs selected (checked) within the Sensitivity Setting **Outputs** tab are shown in the Sensitivity Analysis panel's **Data** tab as columns with a light-yellow background. When a calculation is completed, the output cells are filled with calculated values or the symbol "-" if there is no solution for the output.

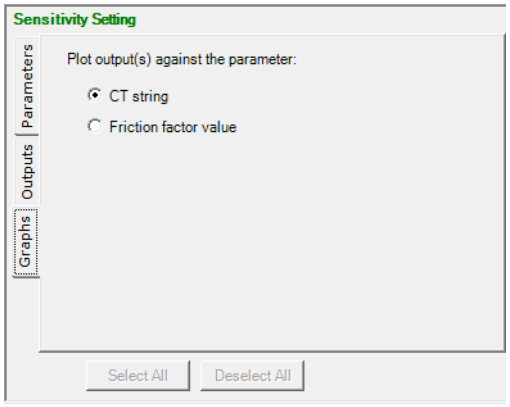
Lockup depth *(CT/JP) or* ***Maximum depth*** *(WL) is always selected and the option is disabled.*

Sensitivity Setting Panel – Outputs Tab



As mentioned previously, **Graphs** are available and visible in the Sensitivity Analysis panel's **Graphs** tab when the number of parameters selected in the Sensitivity Setting panel's **Parameters** tab is one or two. The wizard makes graphs of the selected outputs vs. the primary parameter. If there are two selected parameters, the user may select the primary parameter against which curves are plotted.

Sensitivity Setting Panel – Graphs Tab



Sensitivity Analysis Panel

Use the Sensitivity Analysis panel to edit parameter cell values and to view graphs calculated from the cell values when the number of parameters selected in the Sensitivity Setting panel's **Parameters** tab is one or two.

➢ To edit a numeric parameter cell value

1. Select the cell.

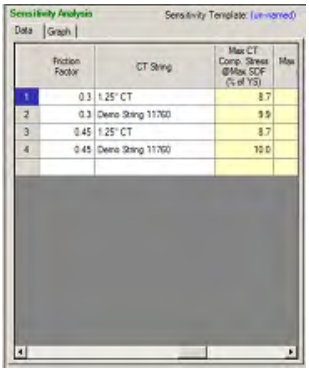
2. Input the value.

➢ To edit a non-numeric value (CT/JP string, wireline, BHA or toolstring, or well)

1. Double-click the cell.

2. Select from the items available in the associated dialog.

Sensitivity Analysis Panel



As noted earlier, graphs are visible in the Sensitivity Analysis panel's **Graphs** tab when the number of parameters selected in the Sensitivity Setting panel's **Parameters** tab is one or two.

Graph type is determined by the type of the primary parameter: curves for a primary parameter of numeric type; bars for a primary parameter of non-numeric type. Legends for curves or bars are displayed to the right of the graph.

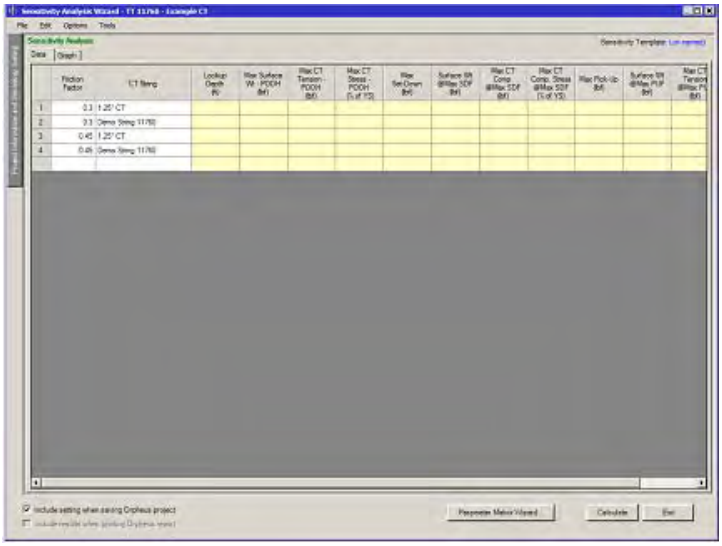
Graph titles (or the outputs) are listed in the dropdown below the graph, and legends are available from which the user may select different views of the graphs.

Panel Viewing Options

When the left-most panels (which includes both Project Information panel and the Sensitivity Setting panel) are visible, they may be hidden or maximized by using the buttons in the top, right-hand portion of the Project Information panel.  When manipulation of the panels causes the buttons to become unavailable, click the **Project Information and Sensitivity Setting** button that appears vertically in the left-most portion of the wizard to restore the left-most panels.

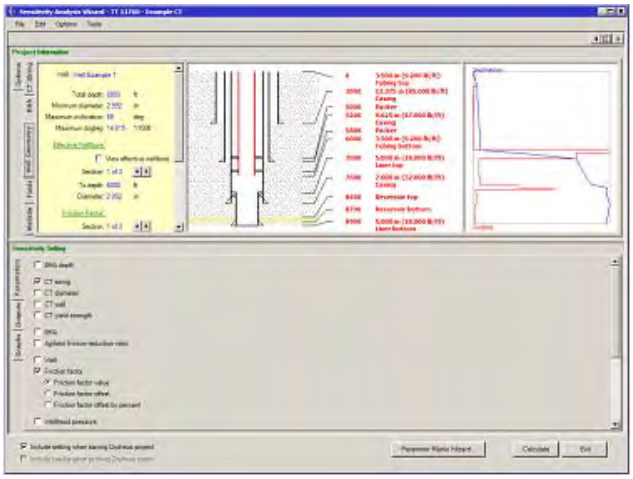
➢ To alter the panel viewing method

1. Click the left-pointing button ◂ to hide the left-side panels and to maximize the Sensitivity Analysis panel's table and graph tabs.



2. Click the **Project Information and Sensitivity Setting** button that appears vertically in the left-most portion of the wizard to restore the previous panel view.

3. Click the right-pointing button ▸ to maximize the left panels and to hide the Sensitivity Analysis panel.



4. Click the middle button ◫ to restore the panel layout to its original configuration.

Bottom Panel: Parameter Matrix Wizard and Calculate Button

The bottom panel contains the **Parameter Matrix Wizard...** button, the **Calculate** button, and the **Exit** [Sensitivity Analysis] button.

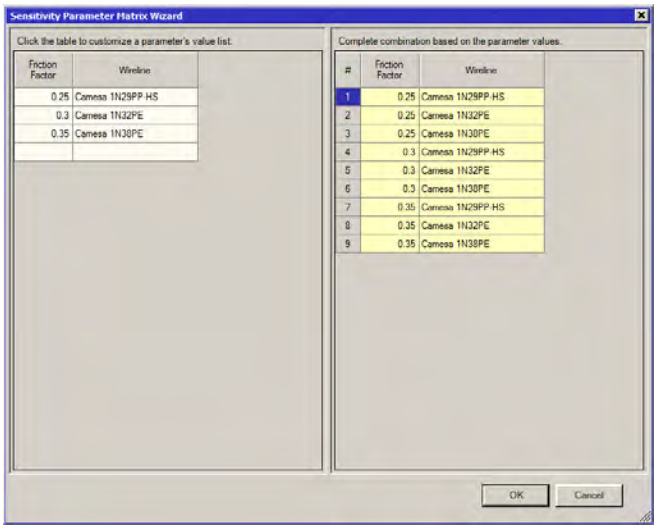
The Parameter Matrix Wizard allows customization of the parameter value lists and the generation of complete combinations of the parameter-value lists, so that the user does not have to prepare a large number of data rows manually (see "Definitions, Overview and Examples" on page 113 for additional information).

To use the Parameter Matrix Wizard

1. Click the **Parameter Matrix Wizard...** button to launch the wizard.

 Lists of parameter values from the main Sensitivity Analysis panel's **Data** tab are loaded in the table on the left-hand side, and the matrix corresponding to the complete combination of the parameter value lists is displayed on the right-hand side.

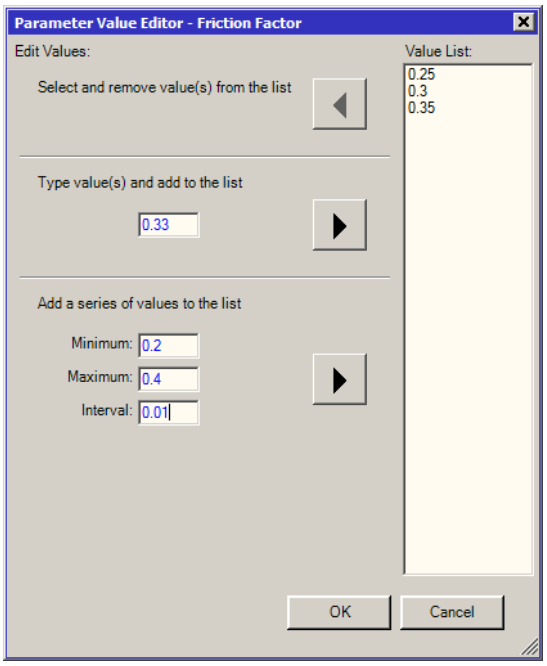
 Parameter Matrix Wizard



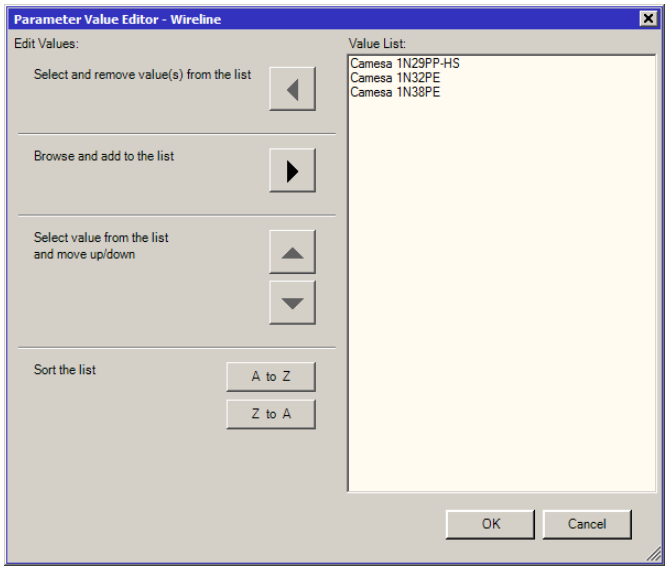
2. To customize the parameter-value list, click a parameter column in the value-list table.

 The Parameter Value Editor dialog appears. The Parameter Value Editor dialog appears in one of two forms: one for numeric parameters and one for non-numeric parameters.

 Numeric Parameter Value Editor



Non-numeric Parameter Value Editor



3. Use the **Edit Values** instructions to edit the parameters.

4. Click **OK** to return to the Sensitivity Parameter Matrix Wizard, where the edited list of values has populated the parameter column (left-hand side), replacing the original list, and the matrix on the right-hand side has been automatically updated.

5. Click **OK** on the Sensitivity Parameter Matrix Wizard to return to the main Sensitivity Analysis Wizard display.

 The parameters in the **Data** tab of the Sensitivity Analysis panel have been replaced with the edited values.

➢ To calculate the outcome of the sensitivity analysis

1. Click the **Calculate** button.

 A progress bar is shown at the top right-hand side of the Sensitivity Analysis Wizard display.

2. Press the **ESC** key or click the link label to cancel the calculation.

Definitions, Overview and Examples

Sensitivity Setting Definitions

Parameter: A parameter is an item to which the user can assign value(s) for the purpose of overriding current values(s) in the current Orpheus project. Parameter(s) are selected by the user and the value(s) are used in the sensitivity calculation.

Output: An output is an item that the wizard calculates. An output is selected by the user.

Primary Parameter: The primary parameter is the parameter against which the calculated output(s) are plotted, and so is the x-axis of the graph.

Secondary Parameter: The secondary parameter corresponds to the different curves of an output vs. the primary parameter. Values of the secondary parameter are displayed as the legends of a graph.

Analysis Table: The sensitivity analysis table is comprised of two parts: the **parameter** columns for the user to enter values and the **output** columns to display the calculated values (these have a light-yellow background color).

Parameter 1	Parameter 2	…	Output 1	Output 2	…

Parameter Matrix: The parameter matrix is comprised of rows of parameter values. Following is an example in which the matrix is the complete combination of three parameter-value lists. However, a matrix does not have to be a complete combination of parameter-value lists.

Parameter-value Lists

Parameter 1	Parameter 2	Parameter 3
P11	P21	P31
P12	P22	P32
		P33

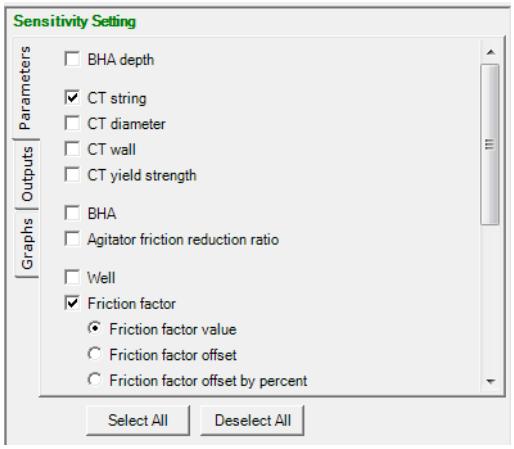
Parameter Matrix

Parameter 1	Parameter 2	Parameter 3
P11	P21	P31
P11	P21	P32
P11	P21	P33
P11	P22	P31
P11	P22	P32
P11	P22	P33
P12	P21	P31
P12	P21	P32
P12	P21	P33
P12	P22	P31
P12	P22	P32
P12	P22	P33

Sensitivity Setting: Information associated with an Orpheus project and saved in an Orpheus database for the purpose of performing a sensitivity analysis. The information includes:

- Selection of parameters
- Selection of outputs
- Primary parameter
- Parameter matrix

Sensitivity Setting Panel in Sensitivity Analysis Wizard



Sensitivity Template: Information for performing a sensitivity analysis, saved in an Orpheus database, but not associated with any Orpheus project. The user may save current sensitivity analysis information as a template by selecting **File > Save as Template** from the menu bar.

Sensitivity Analysis Overview

The sensitivity analysis wizard can be used to calculate selected items (Outputs) using current project data and user-entered data for the selected parameters.

When the number of selected parameters is one or two, the wizard also plots graphs of the selected outputs vs. the primary parameter.

For this version of Cerberus, the available parameters for different application types are shown below:

Sensitivity Analysis Parameters

Parameters of Sensitivity Analysis			
CT	**FLATpak**	**JP**	**WL**
BHA depth	BHA depth	BHA depth	Toolstring depth
CT string	FLATpak	JP string	Wireline
CT diameter		Pipe OD	Cable diameter
CT wall		Pipe ID	
CT yield strength		Pipe yield strength	Cable breaking strength
BHA	BHA	BHA	Toolstring
Agitator friction reduction ratio	Agitator friction reduction ratio	Agitator friction reduction ratio	
Well	Well	Well	Well
Friction factor	Friction factor	Friction factor	Friction factor
Wellhead pressure	Wellhead pressure	Wellhead pressure	Wellhead pressure
Well fluid density	Well fluid density	Well fluid density	Well fluid density
Well liquid flow rate	Well liquid flow rate	Well liquid flow rate	Well liquid flow rate
CT fluid density	CT fluid density	Pipe fluid density	
CT liquid flow rate	CT liquid flow rate	Pipe liquid flow rate	
Force on end - RIH	Force on end - RIH	Force on end - RIH	Force on end - RIH
Force on end - POOH	Force on end - POOH	Force on end - POOH	Force on end - POOH
CT speed	CT speed	Pipe speed	Cable speed
Torque on end		Torque on end	
Surface rotation	Surface rotation	Surface rotation	

The available outputs and the descriptions for different application types are shown below:

Available Sensitivity Analysis Outputs

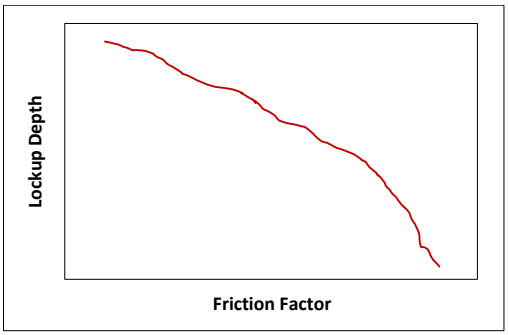
Outputs of Sensitivity Analysis		
CT/FLATpak	**JP**	**WL**
Lockup depth	Lockup depth	Maximum depth
		Minimum surface weight during RIH
		Minimum cable tension during RIH
Maximum surface weight during POOH	Maximum surface weight during POOH	Maximum surface weight during POOH
Maximum CT tension during POOH	Maximum pipe tension during POOH	Maximum cable tension during POOH
Maximum CT stress during POOH (% of BS)	Maximum pipe stress during POOH (% of BS)	Maximum cable stress during POOH (% of BS)
Maximum set-down force	Maximum set-down force	Maximum set-down force
Surface weight @max SDF	Surface weight @max SDF	Surface weight @max SDF
Maximum CT compression @max SDF	Maximum pipe compression @max SDF	
Maximum CT compressive stress @max SDF (% of YS)	Maximum pipe compressive stress @max SDF (% of YS)	
Maximum pick-up force	Maximum pick-up force	Maximum pick-up force
		Maximum over-pull at cablehead
Surface weight @max PUF/OP	Surface weight @max PUF/OP	Surface weight @max PUF/OP
Maximum CT tension @max PUF	Maximum pipe tension @max PUF	Maximum cable tension @max PUF/OP
Maximum CT tensile stress @max PUF (% of YS)	Maximum pipe tensile stress @max PUF (% of YS)	Maximum cable tensile stress @max PUF/OP (% of BS)

Example 1 -- Single Parameter

If :
Friction factor is the only parameter selected,
And a list of **Friction factor** values is entered,
And Lockup depth is the only output selected (as shown below),
Then:
The wizard calculates the lockup depths based on current project data and the friction factor values given here.

Also, a curve of **Lockup depth** vs. **Friction factor** can be plotted to show how sensitive **Lockup depth** is to **Friction factor.**

Friction Factor	Lockup Depth
X1	Y1
X2	Y2
X3	Y3
X4	Y4
X5	Y5
X6	Y6
User Input	Wizard Calculated



Example 2 -- Two Parameters

If :
Friction factor and **CT string** are the parameters selected,
And a matrix of parameter values is entered,
And lockup depth and maximum set-down force are the outputs selected,
Then:
The wizard calculates the lockup depths and maximum set-down forces based on current project data, and the friction factor values and CT strings given here.

Graphs can be plotted for the calculated results. However, the primary parameter, against which the calculated lockup depths and maximum set-down forces are plotted, must be selected first. The primary parameter

also determines the graph type. If the primary parameter is not of numeric type (for example, **CT string**, or **Well**) then bar charts, instead of curves, are plotted.

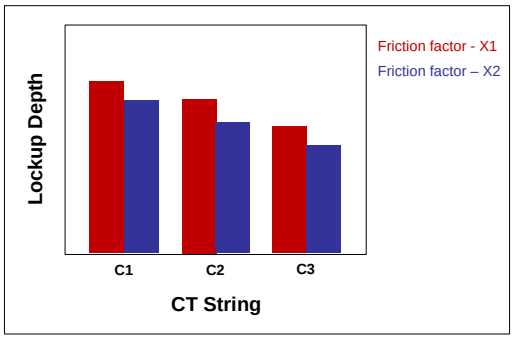
Case 1 – Complete combination of parameter value lists

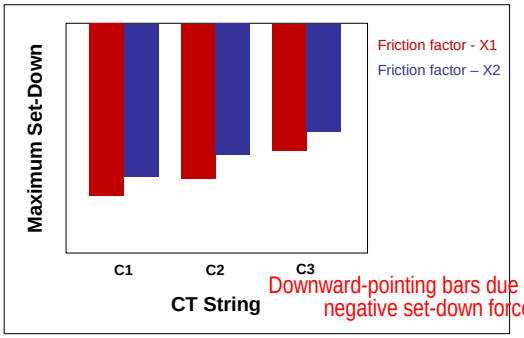
If **CT string** is selected as the primary parameter, then bar charts of lockup depth and maximum set-down force, respectively, vs. CT string, can be plotted, as shown below.

Primary Parameter

Friction Factor	CT String	Lockup Depth	Maximum Set-Down
X1	C1	L11	S11
X1	C2	L12	S12
X1	C3	L13	S13
X2	C1	L21	S21
X2	C2	L22	S22
X2	C3	L23	S23

User Input Wizard Calculated



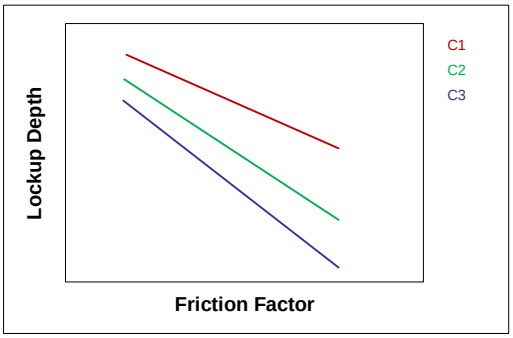


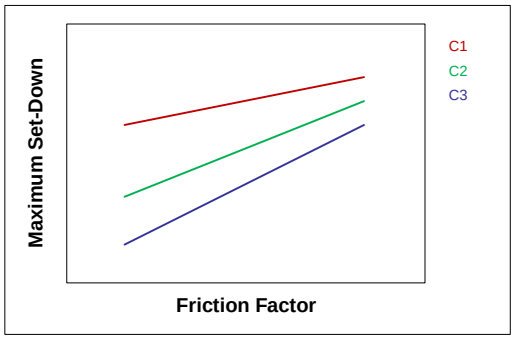
Now, if **Friction factor** is selected as the primary parameter, then curves of lockup depth and maximum set-down force, respectively, vs. friction factor, are plotted as shown below.

Primary Parameter

CT String	Friction Factor	Lockup Depth	Maximum Set-Down
C1	X1	L11	S11
C1	X2	L21	S21
C2	X1	L12	S12
C2	X2	L22	S22
C3	X1	L13	S13
C3	X2	L23	S23

User Input Wizard Calculated





Case 2 – Incomplete combination of parameter value lists

The parameter matrix need not be a complete combination of the selected parameter value lists. For this example, if the 4th and 5th rows of the matrix are missing or if there is no solution for these two rows (displayed as "-"), and assuming the primary parameter is **CT string**, then the table and the graphs appear as follows:

Primary Parameter

Friction Factor	CT String	Lockup Depth	Maximum Set-Down
X1	C1	L11	S11
X1	C3	L13	S13
X2	C1	L21	S21
X2	C2	L22	S22

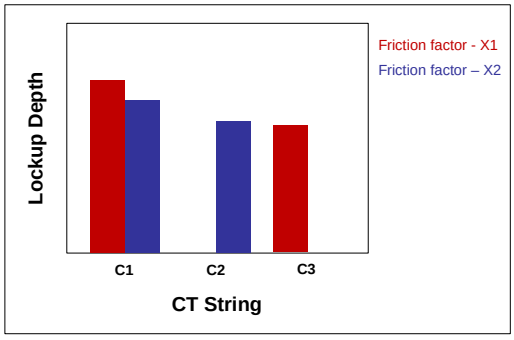
Missing two rows

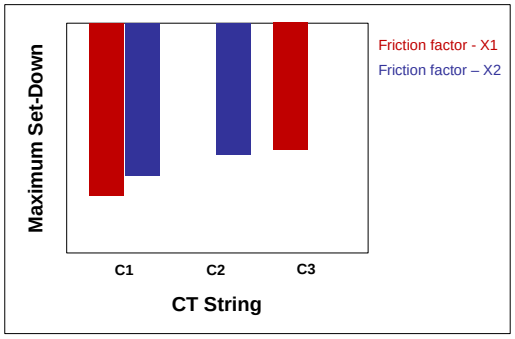
or

Primary Parameter

Friction Factor	CT String	Lockup Depth	Maximum Set-Down
X1	C1	L11	S11
X1	C2	-	-
X1	C3	L13	S13
X2	C1	L21	S21
X2	C2	L22	S22
X2	C3	-	-

No solution (-) for two rows





Now, if the primary parameter is **Friction factor**, then the table and the graphs will appear as follows:

Primary Parameter

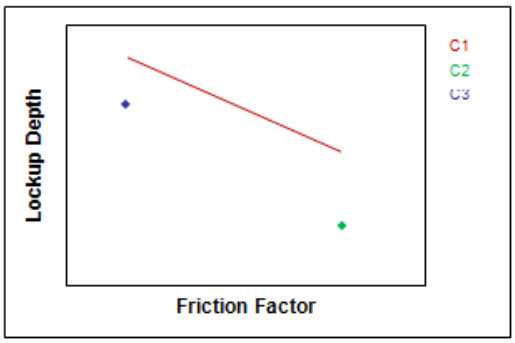
CT String	Friction Factor	Lockup Depth	Maximum Set-Down
C1	X1	L11	S11
C1	X2	L21	S21
C2	X2	L22	S22
C3	X1	L13	S13

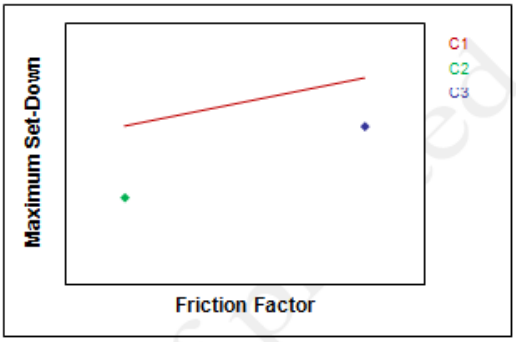
Missing two rows

Primary Parameter

CT String	Friction Factor	Lockup Depth	Maximum Set-Down
C1	X1	L11	S11
C1	X2	L21	S21
C2	X1	-	-
C2	X2	L22	S22
C3	X1	L13	S13
C3	X2	-	-

No solution (-) for two rows





Example 3 – More than Two Parameters

If this situation, the wizard calculates the selected outputs based on current project data and the parameter values given here. No graph will be plotted.

Parameter 1	Parameter 2	...	Output 1	Output 2	...
P11	P21		O11	O21	
P12	P22		O12	O22	
P13	P23		O13	O23	
...	...		...	...	

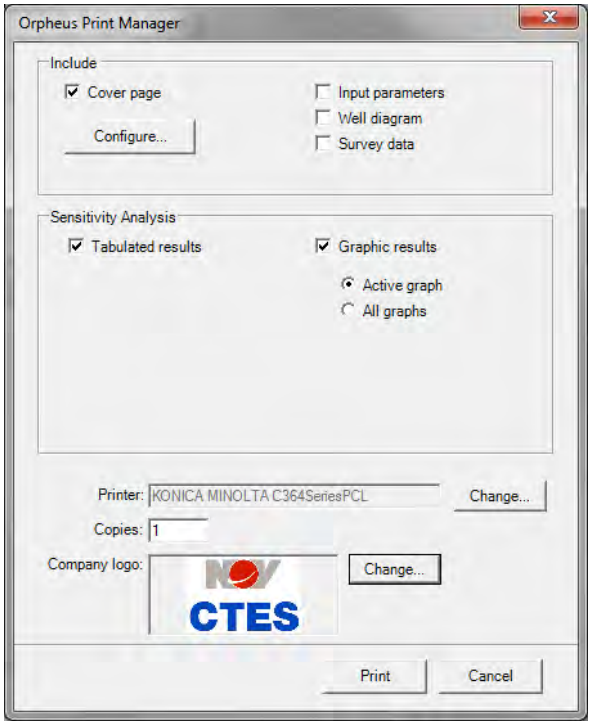
Printing Sensitivity Analysis Results

The user may print the sensitivity analysis results for either inside the Sensitivity Analysis wizard display screen or as an appendix to the main Orpheus report.

➢ To print sensitivity analysis results from the Sensitivity Analysis Wizard

1. From the Sensitivity Analysis Wizard, select **File** > **Print** from the menu bar.

 The Orpheus Print Manager appears.



2. From the Orpheus Print Manager, click (to check) **Tabulated results** and/or **Graphic results** (default is that both are checked).

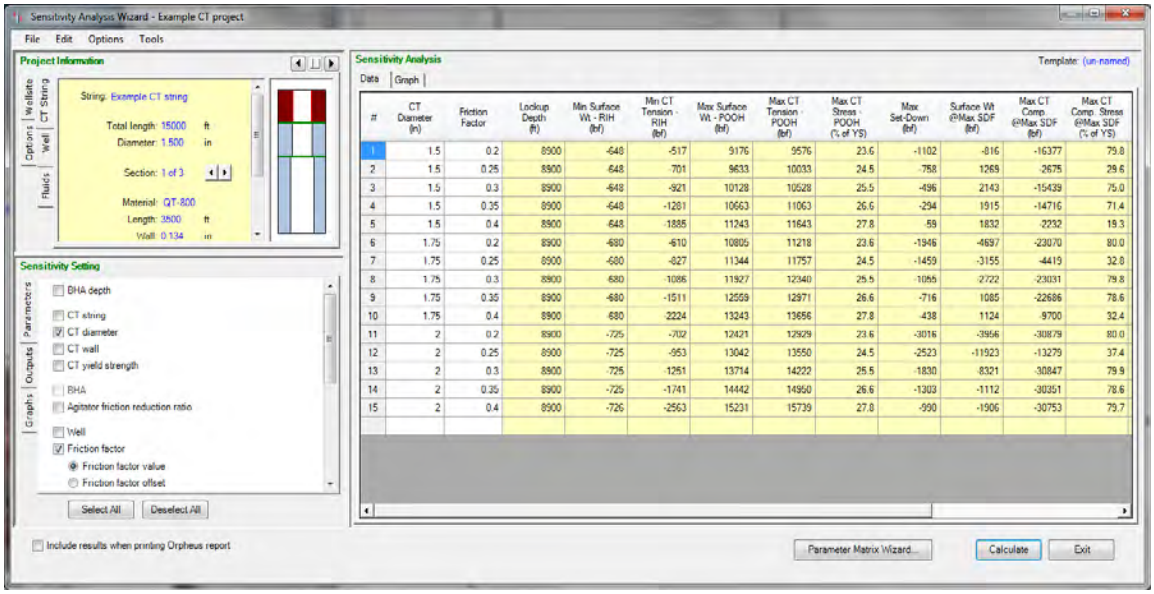
 a. Click (to toggle) **Active graph** or **All Graphs** under **Graphic results**.

3. Select the **Print** button.

➢ To include sensitivity analysis results in the main Orpheus report

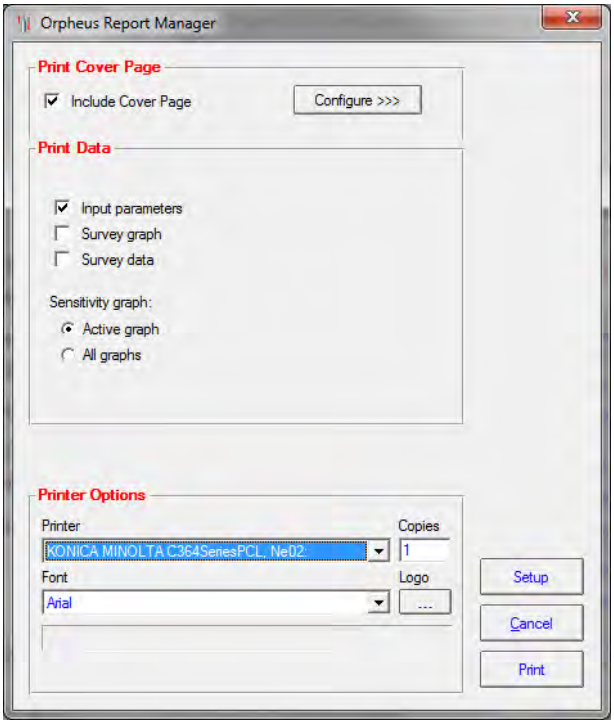
1. From the Sensitivity Analysis display screen, after having completed the calculation, click **Include results when printing Orpheus report** in the lower left-hand corner of the display.

Sensitivity Analysis Calculated



2. From the main Orpheus interface (with the to-be-printed project open), select **File > Print** from the menu bar.

 The Orpheus Report Manager appears.

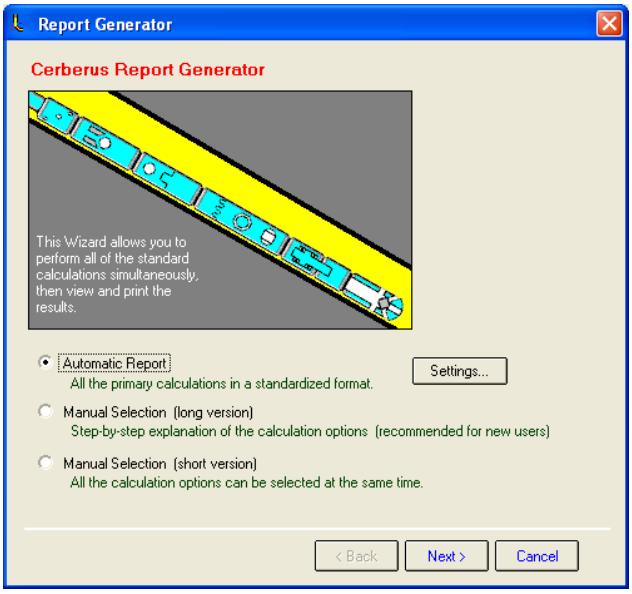


3. [Click to] select from the Sensitivity graph options and click **Print.**

The results are included in the Orpheus report as an appendix.

Report Generator

The Report Wizard performs all of the standard calculations simultaneously. Calculations can then be viewed or printed.

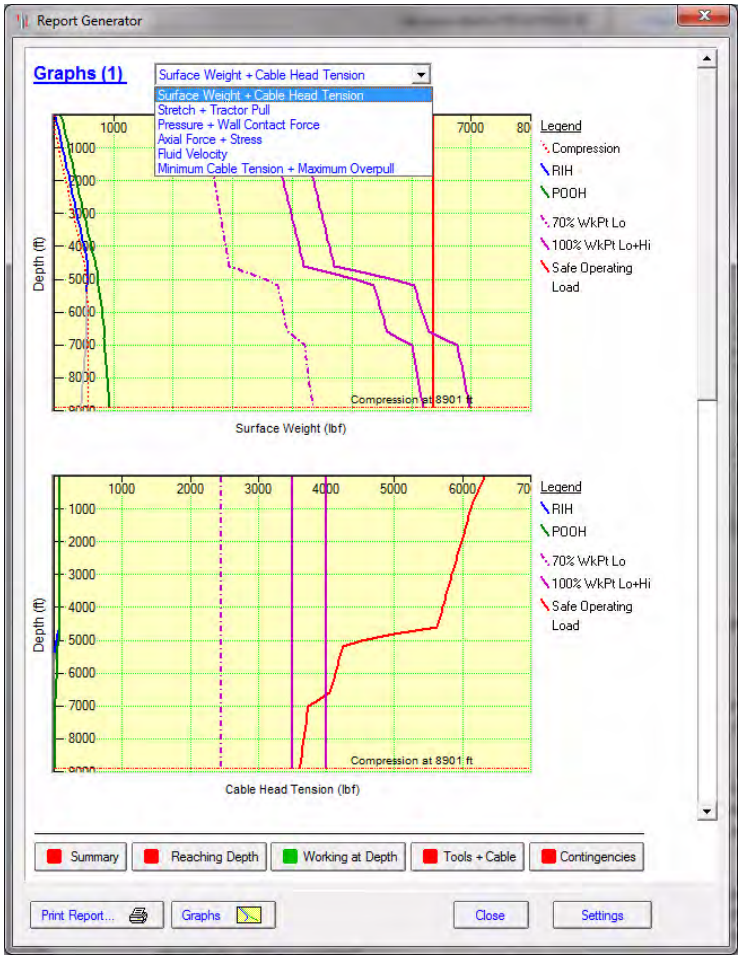


*On the **Q & A** tab, select **Other calculations**, then **Use the Report Generator to create a full analysis and print a report**.*

When Automatic Report (CT only) is toggled in the Report Generator wizard, the resulting generated report defaults to the **Summary** view. Various other views are available (via buttons), including **Reaching Depth**, **Working at Depth**, **Workstring**, and **Contingencies**.

Graphical representations are available via the **Graphs** 

button on a generated report, and the type of graph is available via a dropdown selection at top-of-page.



From the graphical representations, the user may set graph configurations to plot the scale or to zoom out of a zoomed graph.

➢ To set graph configurations

1. Right-click within a graph.

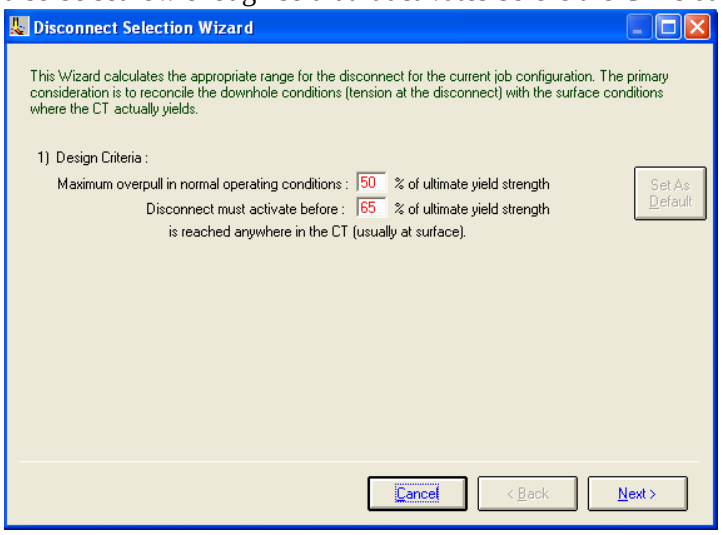
2. Select either **Scale** or **Zoom Out**.

 a. If **Scale** is selected, a Plot Scale popup appears, from which the user may specify **Minimum** and **Maximum Tension** and **Minimum** and **Maximum Depth**.

 b. Selecting **Zoom Out** is effective only if the graph has been zoomed-in using the mouse click-and-drag method on the graph surface.

Disconnect (CT)

The Disconnect Wizard (CT) helps to select the optimum setting for a mechanical disconnect. The mechanical disconnect must be set high enough so that it does not interfere with normal operation; however, it must also be set low enough so that it activates before the CT is damaged.

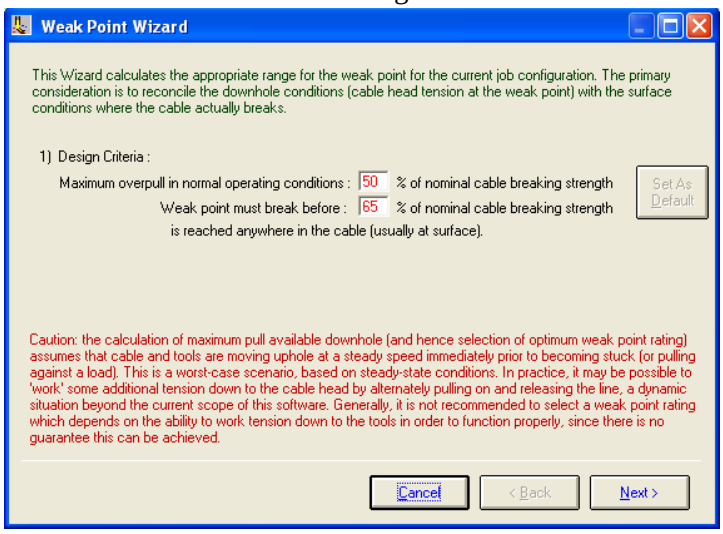


In the Disconnect Wizard, the **Maximum overpull in normal operating conditions**, as well as the **Disconnect must activate before** are specified as percentages of the yield strength. The wizard then calculates the connector tensions for when those percentages are reached anywhere along the CT (usually at surface). Finally, a disconnect setting that is in between the two connector tension-values can be selected.

*On the **Q & A** tab, select **Selecting the CT string and BHA**, then **What is the optimum release setting for a mechanical disconnect?***

Weak Point Selection (WL/JP)

The Weak Point Selection wizard helps to select the optimum weak point for the job. The weak point must be high enough so that it does not interfere with normal operation; however, it must also be low enough so that it breaks before the cable is damaged.



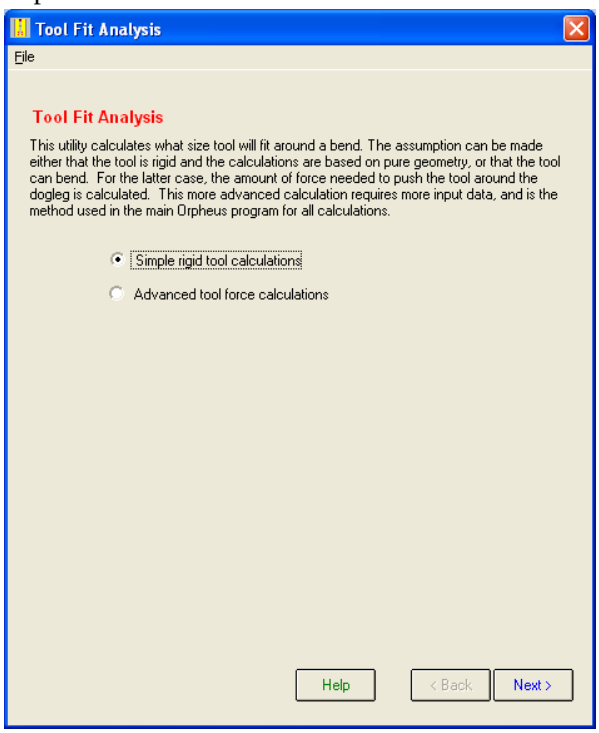
In the Weak Point Wizard, the maximum overpull in normal operations, as well as when the cable must break – as percentages of the nominal cable breaking strength – are specified. The wizard then calculates the cable head tensions for when those percentages are reached anywhere along the cable (usually at surface). Finally, a weak point that is in between the two cable head tension values can be selected.

*On the **Q & A** tab, select **Selecting the toolstring**, then **What is the best weak point for this job?***

Tool Fit Analysis

The Tool Fit Analysis wizard uses the highly sophisticated Finite Element Analysis (FEA) tool-bending model for its calculations.

This utility calculates what size tool will fit around a bend. The assumption can be made either that the tool is rigid and the calculations are based on geometry, or that the tool can bend. In the latter case, the amount of force needed to push the tool around the dogleg is calculated, as are the stresses exerted on the toolstring. This advanced calculation requires additional input data, and is the method used throughout Orpheus.



The FEA model divides the tool into short elements and calculates the position of the tool in the wellbore. The effects of the axial force, variations in curvature, and tool stiffness are included. Special calculations are done for centralizers and knuckle joints and rollers. The resulting drag calculations and stress calculations for the tool are highly accurate.

String Design (CT)

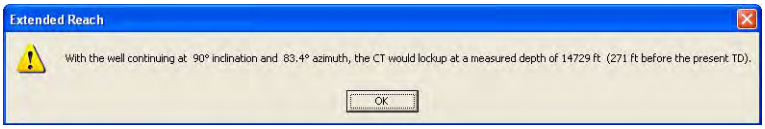
The String Design Wizard helps design a new CT string to meet expected conditions. When the requirements, such as road weight, flow rate, set-down weight, pick-up force, and string manufacturer are specified, Orpheus provides calculations, such as dry weight and estimated costs.



*On the **Q & A** tab, select **Selecting the CT string and BHA,** then **What is the optimum string design for this job**?*

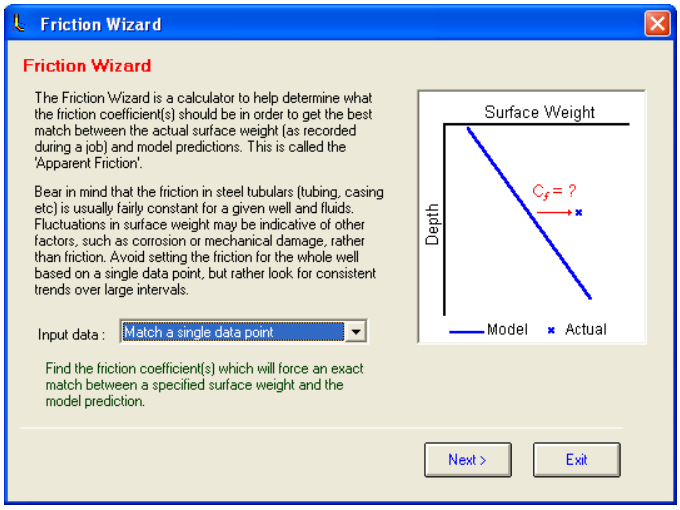
Extended Reach (CT/JP)

The Extended Reach wizard is used to project past the current well TD. This helps to predict how far the user could possibly proceed if the well continued on its present trajectory.



Friction Coefficient Analysis

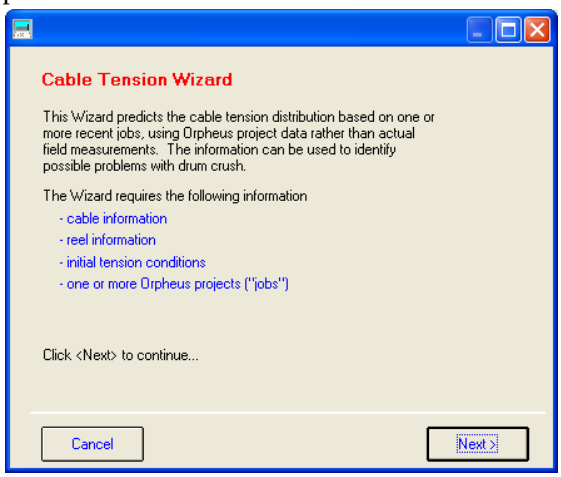
The Friction Wizard is used to help determine what the friction coefficient(s) should be in order to get the best match between the actual surface weight (as recorded during a job) and model predictions. This is called the "Apparent Friction".



Keep in mind that the friction in steel tubulars (tubing, casing, etc.) is usually fairly constant for a given well and fluids. Fluctuations in surface weight may be indicative of other factors, such as corrosion or mechanical damage, rather than friction. Avoid setting the friction for the entire well based on a single data point, but rather look for consistent trends over large intervals.

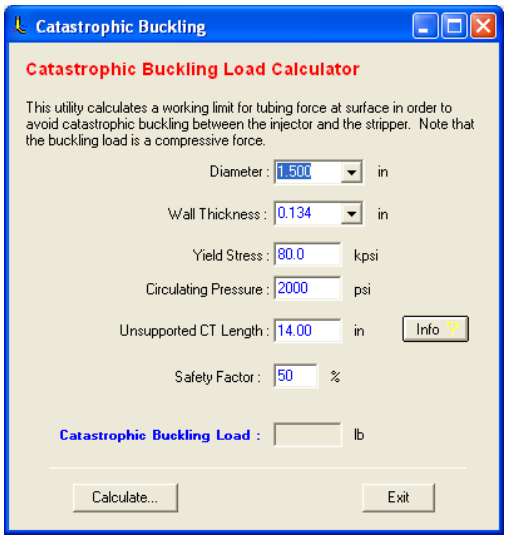
Cable Tension Wizard (WL)

The Cable Tension Wizard predicts the cable tension distribution based on one or more recent jobs, using Orpheus project data rather than actual field measurements. The information can be used to identify possible problems with drum crush.



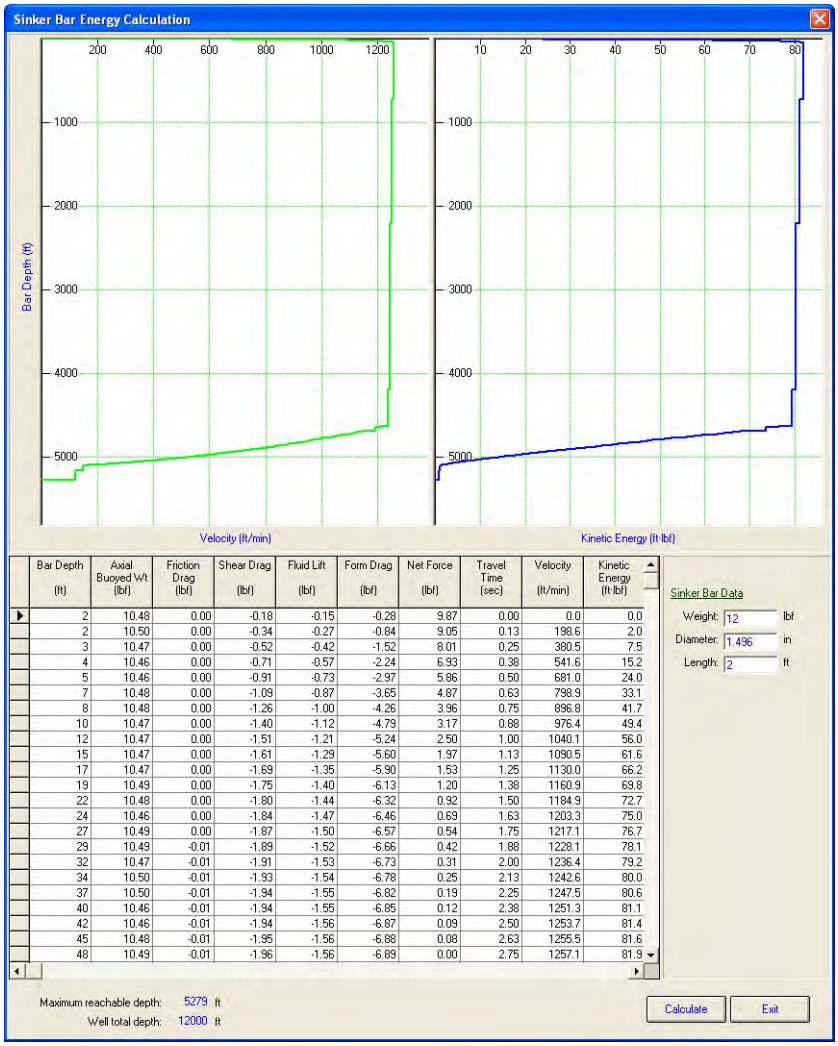
Catastrophic Buckling (CT)

The Catastrophic Buckling Load Calculator utility calculates a working limit for tubing force at surface in order to avoid catastrophic buckling between the injector and the stripper. Note that the buckling load is a compressive force.



Sinker Bar Energy

The Sinker Bar Energy utility calculates the velocity and the energy created (upon impact of a bar dropped in a deviated well) for the purpose of activating a tool downhole.



Bar Depth (ft)	Axial Buoyed Wt (lbf)	Friction Drag (lbf)	Shear Drag (lbf)	Fluid Lift (lbf)	Form Drag (lbf)	Net Force (lbf)	Travel Time (sec)	Velocity (ft/min)	Kinetic Energy (ft-lbf)
2	10.48	0.00	-0.18	-0.15	-0.28	9.87	0.00	0.0	0.0
2	10.50	0.00	-0.34	-0.27	-0.84	9.05	0.13	198.6	2.0
3	10.47	0.00	-0.52	-0.42	-1.52	8.01	0.25	380.5	7.5
4	10.46	0.00	-0.71	-0.57	-2.24	6.93	0.38	541.6	15.2
5	10.46	0.00	-0.91	-0.73	-2.97	5.86	0.50	681.0	24.0
7	10.48	0.00	-1.09	-0.87	-3.65	4.87	0.63	798.9	33.1
8	10.48	0.00	-1.26	-1.00	-4.26	3.96	0.75	896.8	41.7
10	10.47	0.00	-1.40	-1.12	-4.79	3.17	0.88	976.4	49.4
12	10.47	0.00	-1.51	-1.21	-5.24	2.50	1.00	1040.1	56.0
15	10.47	0.00	-1.61	-1.29	-5.60	1.97	1.13	1090.5	61.6
17	10.47	0.00	-1.69	-1.35	-5.90	1.53	1.25	1130.0	66.2
19	10.49	0.00	-1.75	-1.40	-6.13	1.20	1.38	1160.9	69.8
22	10.48	0.00	-1.80	-1.44	-6.32	0.92	1.50	1184.9	72.7
24	10.46	0.00	-1.84	-1.47	-6.46	0.69	1.63	1203.3	75.0
27	10.49	0.00	-1.87	-1.50	-6.57	0.54	1.75	1217.1	76.7
29	10.49	-0.01	-1.89	-1.52	-6.66	0.42	1.88	1228.1	78.1
32	10.47	-0.01	-1.91	-1.53	-6.73	0.31	2.00	1236.4	79.2
34	10.50	-0.01	-1.93	-1.54	-6.78	0.25	2.13	1242.6	80.0
37	10.50	-0.01	-1.94	-1.55	-6.82	0.19	2.25	1247.5	80.6
40	10.46	-0.01	-1.94	-1.55	-6.85	0.12	2.38	1251.3	81.1
42	10.46	-0.01	-1.94	-1.56	-6.87	0.09	2.50	1253.7	81.4
45	10.48	-0.01	-1.95	-1.56	-6.88	0.08	2.63	1255.5	81.6
48	10.49	-0.01	-1.96	-1.56	-6.89	0.00	2.75	1257.1	81.9

Pump Down Rate (WL)

This utility/wizard provides the user with the option of entering iterated/varying data for wellhead pressure (WHP) and pump rate, which Cerberus uses in pump down calculations to help find the minimum pump rate required to reach target depth.

If the user does not opt to input iterated data, the software calculates pumpdown rate using the static WHP number specified in the main Orpheus wireline-project interface.

NOV CTES recommends using the varying-data-entry method for enhanced accuracy of the calculated outcome.

➢ To calculate pump down rates allowing varying WHP and pump flow-rate data

1. From a wireline project in Orpheus, select the **Calculate** tab; then select **Calculation Tools** > **Pump Down Rate**.

 The Pump Down Wizard appears.

2. Make changes to the **Tool Depth Range** and/or **Pump Rate Range**, if desired.

3. From the I**nput Parameters** tab, click (to check) the **Use WHP vs Pump Rate data table** togglebox (default).

 The data table provides the user the ability to input **Pump Rate** and **WHP** parameters.

 a. If no data for pressure response to flowrate is available, click (to uncheck) the **Use WHP vs Pump Rate data table** togglebox. All related data input fields become unavailable. During calculation, then, the software calculates pumpdown rate using the static WHP number specified in the main Orpheus wireline-project interface.

4. Input data in the Pump Rate (gal/min) and WHP (psi) area.

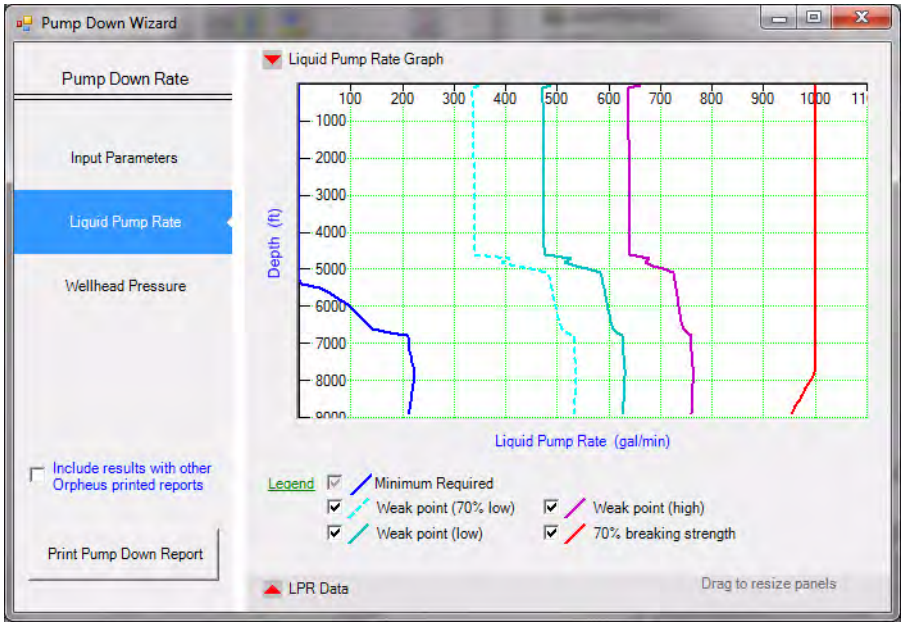
 a. A new row appears when the user places the cursor in the existing bottom row.

 b. Select a row and click **Delete** to delete a row.

 c. Click **Graph** to view a representation of the data.

 d. Select the **Liquid Pump Rate** or **Wellhead Pressure** tab to calculate respective data.

Calculated Limit Pump Rates -- WHP Varied with Pump Rate



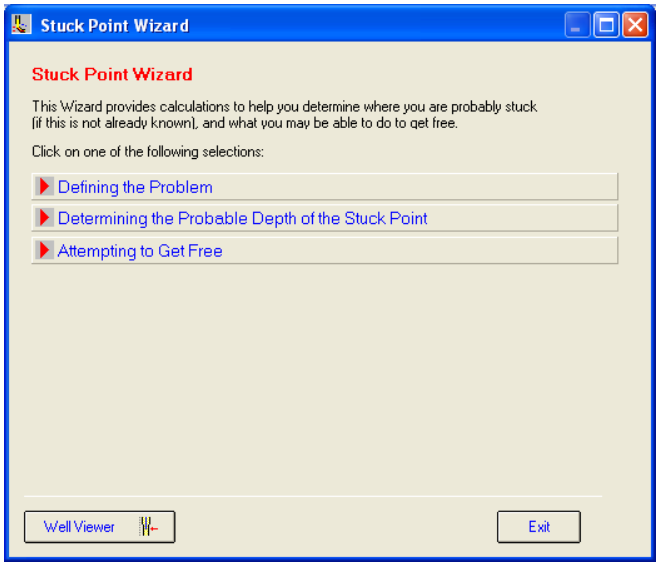
➢ To include the calculated results in the printed report from Orpheus' report generator

1. Proceed with calculations as described previously.

2. Click **Include results with other Orpheus printed report**s.

Stuck Point Analysis

If the CT and/or tool are stuck in the well, the stuck point depth can be calculated and a solution to free the tool determined.

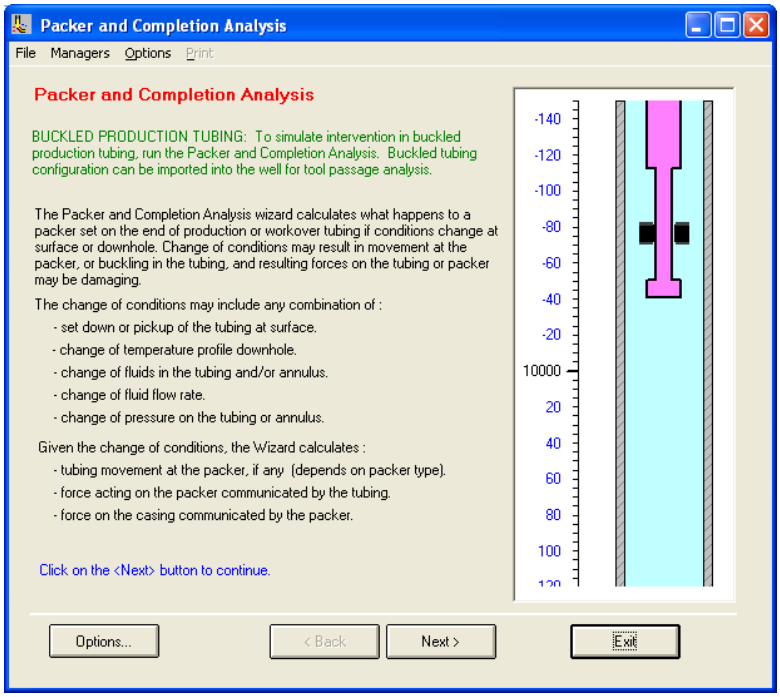


The stuck point calculation uses the gooseneck radius input in the preferences.

The stripper friction can have a considerable effect on the stuck point calculation.

Packer & Completion Analysis

The Packer and Completion Analysis wizard calculates what happens to a packer set on the end of production or workover tubing if conditions change surface or downhole.



Printing Orpheus Project Data

Print the configuration for a project, as well as calculated data.

The data must be calculated before it can be printed.

➢ To Print a Project

1. Open the project in Orpheus.

 The Orpheus main screen appears with the project information loaded.

2. Click **Print** () on the toolbar.

 The Orpheus Report Manager display screen appears.

3. Select the items to print.

 Options to print include a Cover Page, Input Parameters, Survey Graph, and Survey Data.

4. Select the **Printer Options**.

 Choose the printer, the number of copies, the font, and a logo, if not previously selected.

 Any printer connected to the computer, either directly or through a network, can be selected.

5. Click **Print**.

In addition to printing to a printer, a PDF file can be printed for email distribution or archival.

To view a PDF file, the free Adobe Reader must be installed on the computer. It can be downloaded from Adobe's Web site at www.adobe.com.

NOV CTES does NOT recommend the use of any other PDF generator other than the CTES PDF Generator. Use of any other PDF generator can produce unpredictable and undesirable results.

➢ To use the CTES PDF Generator

1. Click **Print** () on the toolbar or click **File** > **Print...** from the menu.

 The Orpheus Report Manager screen appears.

2. Select the items to print.

 Options to print include a Cover Page, Input Parameters, Survey Graph, and Survey Data.

3. Select the Printer Options.

 Choose the number of copies, the font, and a logo, if not previously selected.

4. Select the printer.

 In the Printer Options section, select CTES PDF Printer 25 from the Printer drop-down menu.

5. Click **Print**.

 The Save As dialog box appears.

6. Enter a file name.

 Navigate to the desired location where the file should be saved.

 Ensure the Save as type selection is listed as PDF files (*.pdf). Otherwise, the file might not be saved correctly and might be unavailable for use.

Emailing Project Data

Orpheus data files can easily be exported and emailed to other Cerberus users.

This procedure assumes the user has Microsoft's Outlook email program installed; the steps described are geared toward that end. If the user has a different email program, the steps require modification.

➢ To email job data files

1. Select a project to email.

2. Click **File** > **E-mail Project To...**.

 The Export Orpheus window appears.

3. Select the desired project data file to export.

 All available project data files are listed.

4. Click **OK**.

 The Outlook Address Book window appears.

5. Select or enter the desired recipients of the data file.

 If the recipient is in the address book, then select the recipient from the list. Otherwise, enter an email address for the recipient in the To field.

6. Click **OK**.

 If an Outlook warning appears stating that another program is trying to access the email addresses in Outlook, click Yes to allow the access.

 The email message appears addressed to the selected recipient(s) and with the data file attached.

7. Click **Send**.

Chapter 6
Using Hydra

This section describes the process involved in using the Hydra wellbore hydraulics simulator, the Cerberus hydraulics model for coiled tubing and jointed pipe.

About Hydra

The Cerberus Hydra wellbore hydraulics simulator model performs hydraulics calculations for coiled tubing (CT) and jointed pipe (JP). Included in the model are a large number of tools to help design/model CT and JP pumping operations. Options range from basic “snapshot” calculations to more complex simulations where conditions change with time. Hydra wizards help the user to step through the complex process of choosing fluids and specifying volumes and flow rates.



Hydra supports a wide range of fluid types and models, including gases, foams, and multiphase fluids.

Hydra incorporates the following:

- Power Law, Bingham Plastic, Herschel Bulkley, and Newtonian fluid models
- Foam, gas, and multiphase fluid models
- Well unloading and gas lift design wizards
- Fill Removal wizard, for sand cleanouts
- Velocity String Design wizard
- Pump wizard for circulation volumes and times
- Job Design wizard, for rapid configuration of multi-stage jobs
- “Stage Table” data-entry for time-based simulations
- Reservoir inflow/outflow calculations
- Nozzle and choke calculations

The Hydra Process

The process involved in using Hydra is as follows:

1. Create a new Hydra project or select an existing project (see page 141).

 When creating a new project, choose from two types of project:

 a. Basic Hydraulics involving general circulation of fluids.

 b. Advanced Job Design involving using a design wizard to model a specific job type.

2. Determine a conveyance method (see page 145).
3. (Optional) Create a tool string or select/edit an existing tool string(see page 146).
4. Create a well or select/edit an existing well (see page 147).
5. Select fluids in the job (see page 148).
6. Set initial fluid distribution parameters (see page 148).

7. (Optional) Set advanced project options (see page 151).

8. Perform calculations using application wizards, standard/snapshot calculations, and/or stage tables (see page 151).

Standard Snapshot	Stage Table
a. Set Parameters	a. Set up stage table
b. Calculate	b. Calculate
c. View results and print for hardcopy or electronic transmission	c. View results and print for hard-copy or electronic transmission



Hydra (along with the other Cerberus models) is organized by projects.

Accessing Hydra

➢ To access Hydra from the Cerberus home page

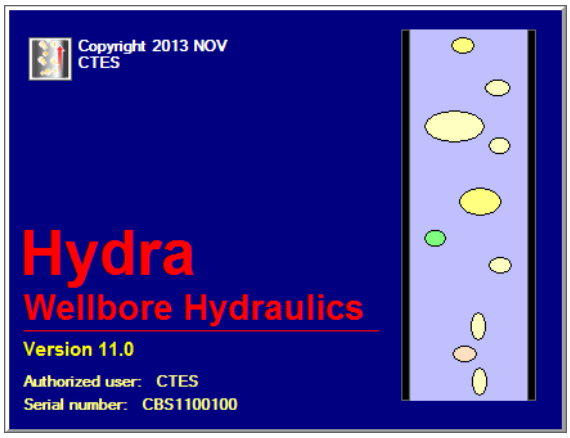
- Select the Hydra button.



OR

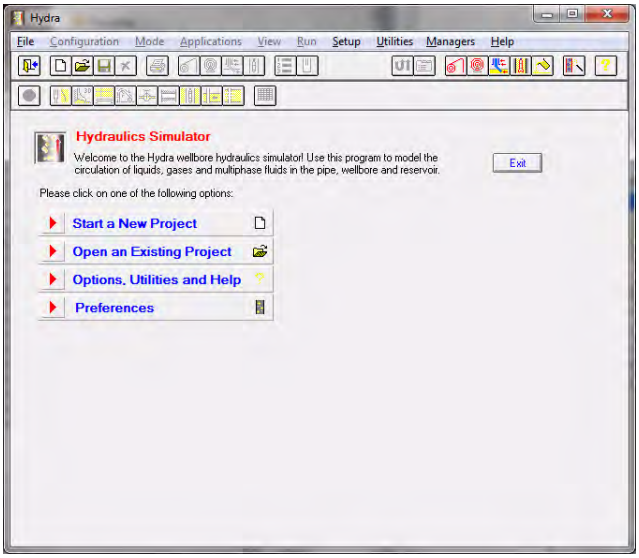
- Select **Models > Hydra** from the menu bar.

 The Hydra model loads, as indicated by the following display screen:



The opening dialog of the Hydra Wellbore Hydraulics model appears.

Hydra Opening Dialog



This Hydra Hydraulics Simulator opening dialog contains the Setup Wizard, which is used to model the circulation of liquids, gases, and multiphase fluids in the pipe, wellbore, and reservoir.

Users may also access the Setup Wizard via the opening-dialog toolbar: . (see page 139).

Hydra's Opening-dialog Tools

Hydra's opening-dialog toolbar contains the following tools:

Hydra Opening-dialog Toolbar

Tool Button	Tool Name	Function
	Exit	Exits Hydra.
	New	Opens the dialog box to create a new Hydra project.
	Open	Opens the dialog box to open an existing Hydra project.
	Save	Saves the current project to disk.
	Delete	Deletes the current project from the hard drive.
	Print	Opens the dialog box to print the current project.
	Select String	Opens the dialog box to select the string.
	Select Reel	Opens the dialog box to select the reel.
	Select Well	Opens the dialog box to select the well.
	Select Tool	Opens the dialog box to select the tool string.
	Select Fluids	Opens the dialog box to select the fluids for the project and their colors.
	Initial Fluid Distribution	Opens the dialog box to select the initial fluid distribution.
	Calculation Setup	Opens the calculation setup dialog box to select the flow path and the inputs for the project.
	Options	Displays the dialog box to set Hydra options (if in Stage Table mode).

Hydra Opening-dialog Toolbar

Tool Button	Tool Name	Function
	String Editor	Starts String Editor with the current string loaded.
	Reel Manager	Starts Reel Manager with the current reel loaded.
	Well Editor	Starts Well Editor with the current well loaded.
	Tool String Editor	Starts Tool Manager with the current tool string loaded.
	Fluids Manager	Starts Fluids Manager.
	Setup Wizard	Starts the Setup Wizard for a step-by-step guide through setting up and starting a new project.
	Help	Provides Hydra-related help.
	Stop Calculations	Cancels a calculation.
	View Well	Zooms in on the well view.
	3D Survey Data	Shows the three-dimensional survey data.
	View Surface	Zooms in on the surface and shows pressures or geometry.
	View Choke	Zooms in on the choke, so the choke inputs can be specified.
	View Reel	Zooms in on the reel, displaying the movement of liquids.
	View Tool	Zooms in on the tool.
	View Reservoir	Zooms in on the reservoir.
	View Perforations	Zooms in on perforations.

Hydra Opening-dialog Toolbar

Tool Button	Tool Name	Function
	Data Table	View data in table format.

Working with Projects

The Hydra model is organized by projects.

CT projects and jointed pipe projects are configured similarly. Calculations are also performed similarly. The main differences between the two types of projects are as follows:

- CT projects use a string of CT; jointed pipe projects use a string of JP.
- CT projects involve a reel and gooseneck; jointed pipe projects do not.

Creating a New Hydra Project

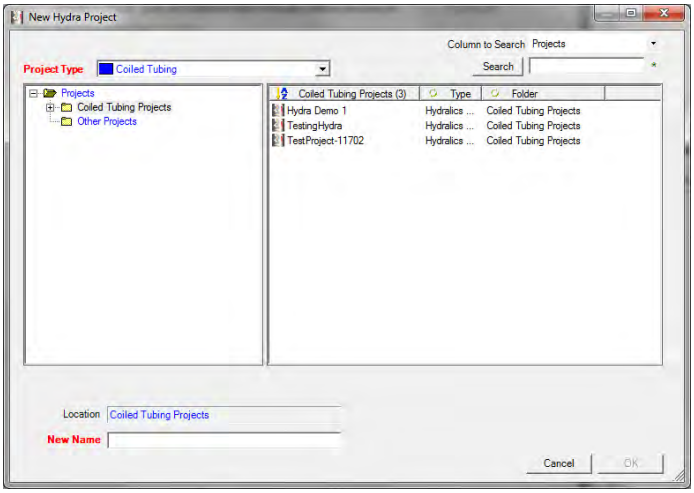
The following procedure outlines how to create a new Hydra project.

➢ To start a new project

1. From the Hydra Opening Dialog, choose one of the following:
 - Select **File > New Project** from the menu bar.
 - Click the **New** toolbar button: 
 - Click the **Start a New Project** button in the Setup Wizard. See “Using the Job Design Wizard” on page 158.

 If either of the first two options is chosen, the **New Project** dialog appears:

New Hydra Project dialog

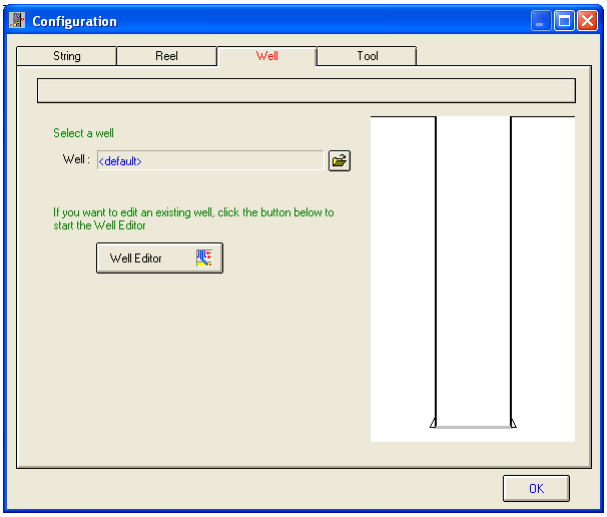


Start a new Hydra project directly from the Cerberus home page by selecting **File > New > Hydraulics Project (Hydra)**.

2. Select the location into which the project is added.

3. Provide a new name for the project. The name may consist of letters, numbers, and spaces, with a maximum of 32 characters.

 The Configuration display screen appears.



4. Configure the project (see Entering Project-configuration Information for more information).

➢ To start a new project using the Setup Wizard

1. Launch the wizard by clicking the Setup Wizard button on the toolbar, or by clicking the **Setup Wizard...** button on the Config tab.

2. Click the **Start a New Project** menu.

3. To model a job with a general circulation of fluids, select **Basic Hydraulics** and do one of the following:

- Click the **Standard** mode button  for basic calculations with steady-state conditions (where time is not a variable).
- Click the **Stage Table** mode button  to specify a fluids and a pump schedule in order to create charts of pressure and other parameters against time.
- Click the **Control Panel** mode button  to set flowrates from a simulated control console (used mostly for training and demonstrations).

OR

To model a specific job type using design wizards, click **Advanced Job Design** and choose one of the following wizards. When the project has been configured, the selected wizard will launch to complete the process.

- Fill Removal Wizard
- Multiphase Wizard
- Well Unloading Wizard
- Foam Wizard
- Velocity String

Where applicable, choose the **Simple Mode** to quickly select or edit a module or choose **Advanced Mode** to model a specific job type using a design wizard.

To . . .	Do this . . .
Show the Setup Wizard automatically when starting Hydra.	Start the Setup Wizard. Select the **Automatically show this Wizard on Startup** checkbox in the lower left corner under the **Preferences** menu.
Stop the Setup Wizard from starting automatically.	Start the Setup Wizard. Disable the **Automatically show this Wizard on Startup** check box in the lower left corner under the **Preferences** menu.
Specify units	Click **Units** under the **Options, Utilities and Help** menu.
Change preferences in Hydra.	From the menu bar, select **Setup > Preferences**.

Opening an Existing Hydra Project

The following procedure describes how to open an existing project.

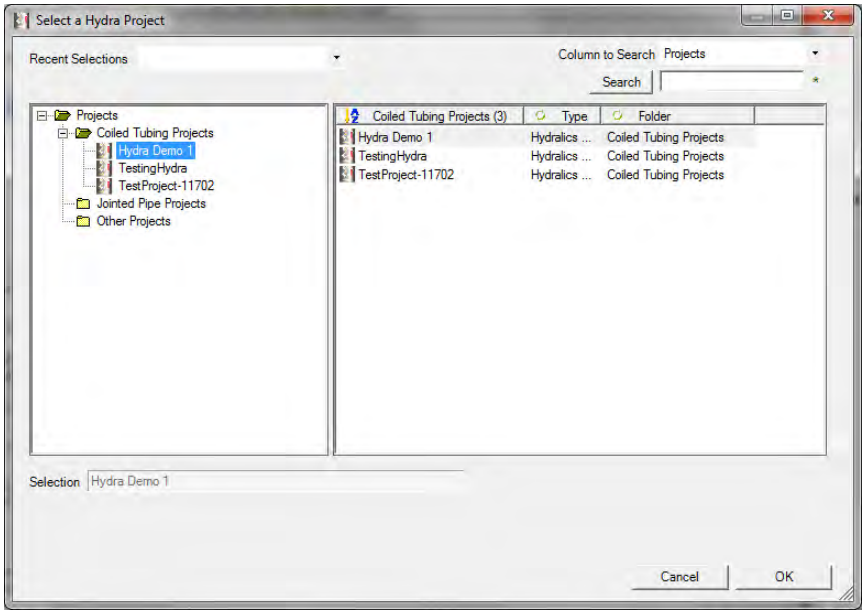
➢ To open an existing project

1. From the Hydra Opening Dialog, choose one of the following:
 - Select **File** > **Open Project** from the menu bar
 - Click the **Open Project** toolbar button: 
 - Click the **Open an Existing Project** button on the opening dialog

 OR

 - Click the **Open an Existing Project** button in the Setup Wizard.

2. The Select a Hydra Project dialog appears, from which to select the desired project.



Organizing Projects into Folders

Hydra is designed with default folders for each type of project (Coiled Tubing Projects; Jointed Pipe Projects). The default folders cannot be renamed, moved, or deleted.

When a new project is created, a folder for the project's placement is automatically selected.

These folders are internal to Cerberus, and unrelated to folders in Windows Explorer.

Projects or folders cannot be placed under different project types. (For example, CT projects cannot be placed under the Jointed Pipe Projects folder.)

Saving a Hydra Project

The project configuration and last set of calculated values is retained when the project is saved. There are two ways a project can be saved:

- Click **Save** () on the toolbar.

 OR

- On the menu bar, click **File** > **Save** or **Save As...**.

Deleting a Hydra Project

When a project is deleted, the information is not recoverable.

➢ To Delete a Project

- On the menu bar, select **File** > **Delete Project**.

Importing a Hydra Project

Import any project that is on removable media into Hydra.

➢ To Import a Project

1. If importing from removable media, insert the media in the drive.
2. Start **Hydra**.

 The Hydra display screen appears.
3. From the menu bar on the Hydra Opening Dialog, select **File** > **Import Project...**.

Exporting a Hydra Project

Export any project in Hydra to removable media for importing to another computer with Hydra.

➢ To Export a Project

1. Start **Hydra**.

 The Hydra Well Intervention Modeling display screen appears.
2. On the menu bar, select **File** > **Export Project...**.

Entering Project-configuration Information

Configuration information is stored within a project, so it is quickly accessible without the necessity of re-input.

1. Select a conveyance string.

 ➢ To select a conveyance string

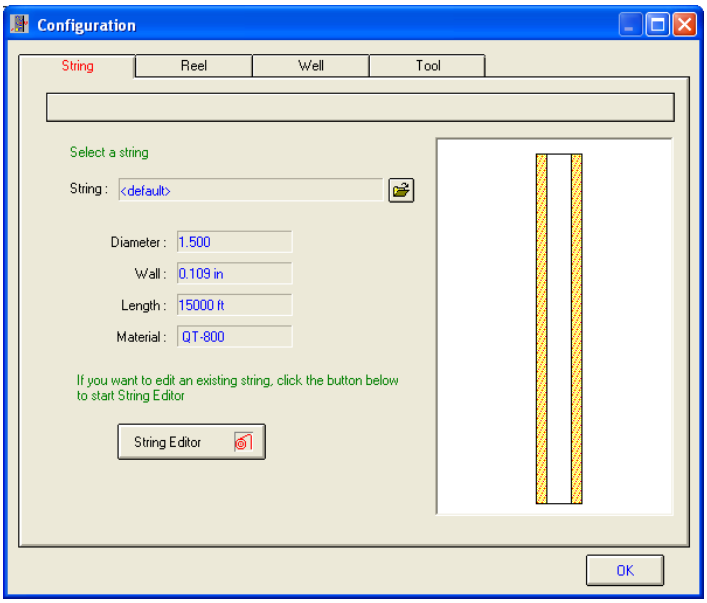
 - From the Configuration display screen (available after providing a name for the new project, or available from the menu-bar item **Configuration** > **String**) complete the following:

Click the yellow-folder to browse and select from existing CT strings or JP strings.

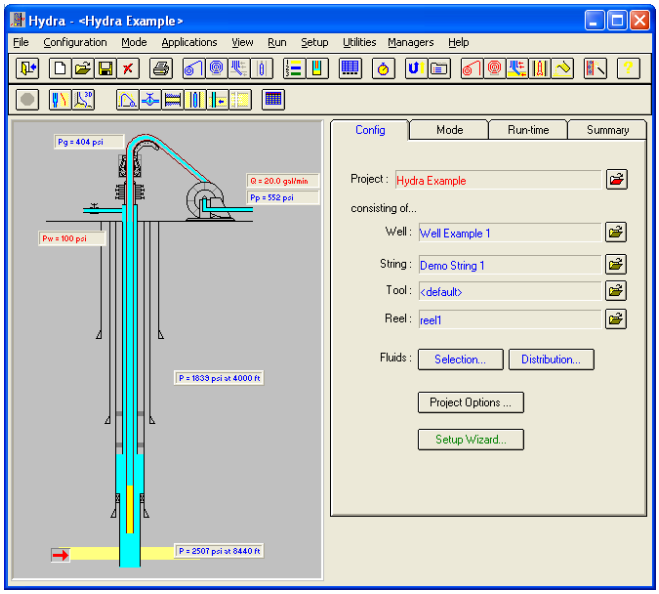
OR

Click the **String Editor** button to edit an existing string.

Hydra Configuration Display Screen



- From the main Hydra display screen, click the yellow-folder icon to browse and select from existing CT Strings or JP Strings.

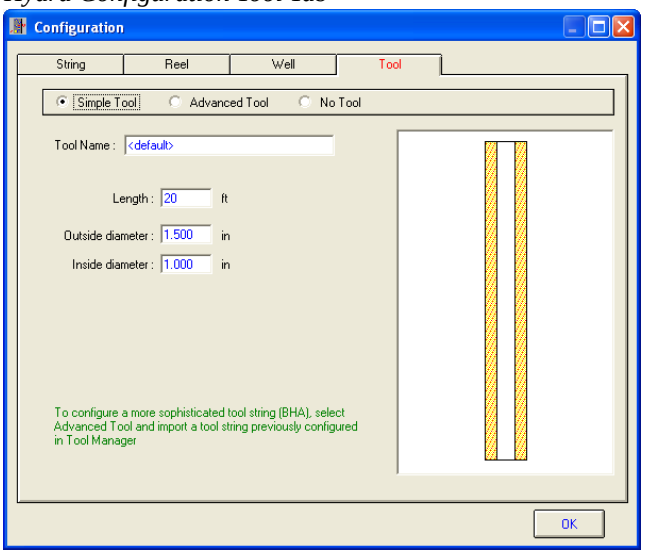


2. (Optional) Configure a tool string.

➢ To configure a tool string

- From the Configuration display-screen **Tool** tab, select **Simple Tool, Advanced Tool**, or **No Tool**.
- To configure a sophisticated tool string (BHA), select **Advanced Tool** and import a tool string previously configured in Tool String Editor.

Hydra Configuration Tool Tab



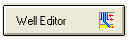
3. Select a well.

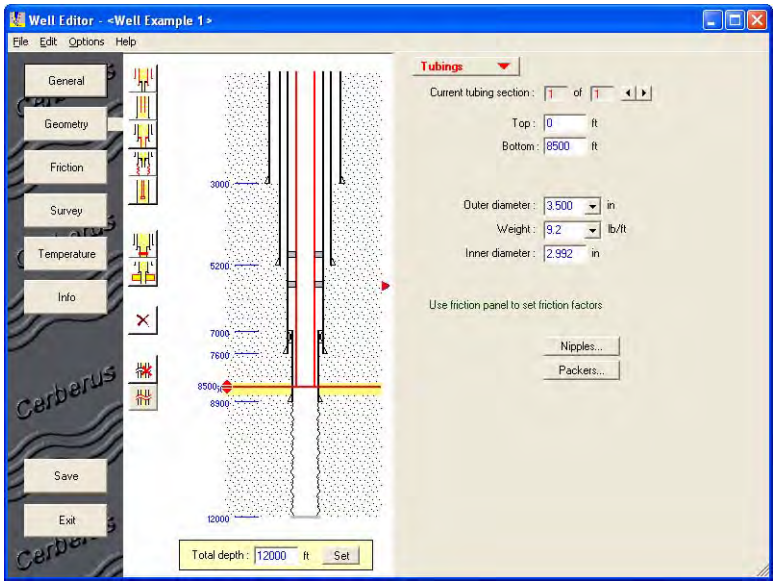
➢ To select a well

a. From the Configuration display screen (available after providing a name for the new project, or available from the menu-bar item **Configuration > Well**), complete the following:

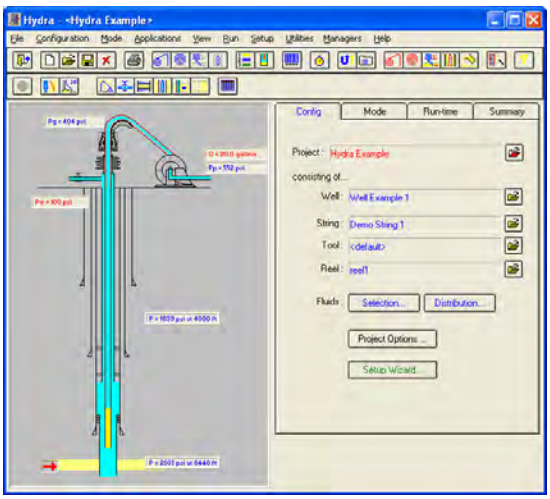
- Click the yellow-folder icon to browse and select from existing wells.

OR

- Click the Well Editor button () to access the Select Well display screen.



b. From the main Hydra display screen, click the yellow-folder icon to browse and select from existing wells.



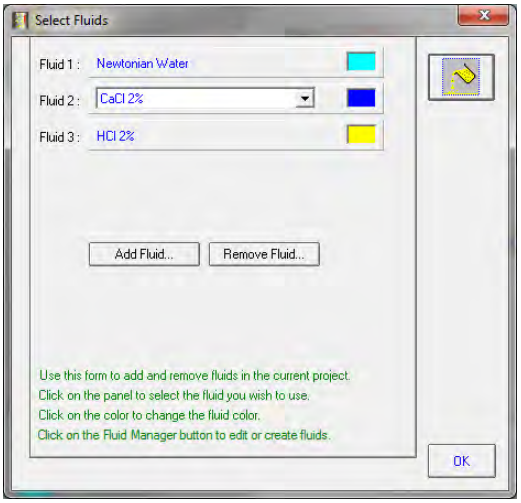
4. Select fluids in the job.

➢ To select fluids in a job

a. From the Hydra display-screen **Config** (default) tab, click the **Selection** button associated with **Fluids**.



The Select Fluids display screen appears.



b. Use the dropdown(s) to select fluids to be added or removed from the project.

c. Click the color associated with the selected fluid to change fluid color.

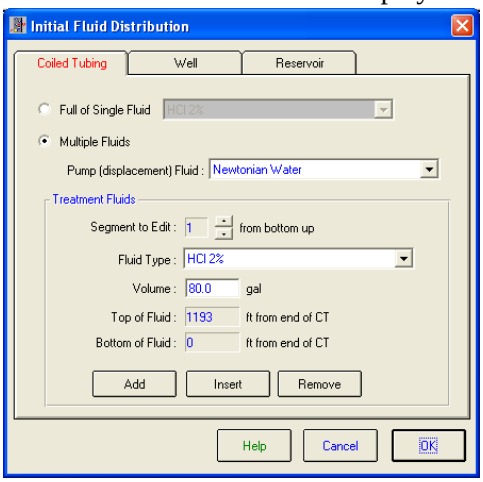
5. Set initial fluid distribution.

➢ To set initial fluid distribution

a. From the Hydra display-screen Config tab, click the **Distribution** button associated with Fluids.



The Initial Fluid Distribution display screen appears.



b. Enter information so that Hydra "understands" the fluid or fluids initially present in the conveyance and the well. The parameters entered here act as a starting point for calculations. (Use the **Help** button on this display screen for more information.)

6. (Optional) Mix pumped and reservoir fluids automatically in the annulus.

When a reservoir is producing and a fluid is pumped, the fluid in the annulus is a mixture of the pumped fluid and the fluid produced from the reservoir. In most cases, the fluid mixture in the annulus is a multiphase fluid whose properties, such as gas-liquid ratio (GLR), are dependent on both reservoir production rate and pump rate. Prior to Cerberus version 10.5, users had to estimate the fluid properties of the annulus mixture and define the fluid mixture in the Fluid Manager.

Generally, a manual iteration scheme has to be used, because the GLR of the annulus mixture depends on the reservoir production rate – but the reservoir production rate is dependent on the flowing bottomhole pressure, which is a function of the annulus fluid properties, including GLR.

This function is only available when a reservoir is included in the calculation. This is because this option is relevant only when the reservoir is producing. When there is no reservoir or when the reservoir is not included, there is no issue of mixing fluids between the pumped fluid and the reservoir fluid.

➢ To include a reservoir in the calculation

1. From the Hydra menu bar, select **Setup** > **Project Options**. (See "(Optional) Set advanced project options.")

OR

Click the **View Reservoir** toolbar button.

2. Click to toggle **Include reservoir properties in calculations**; click **OK**.

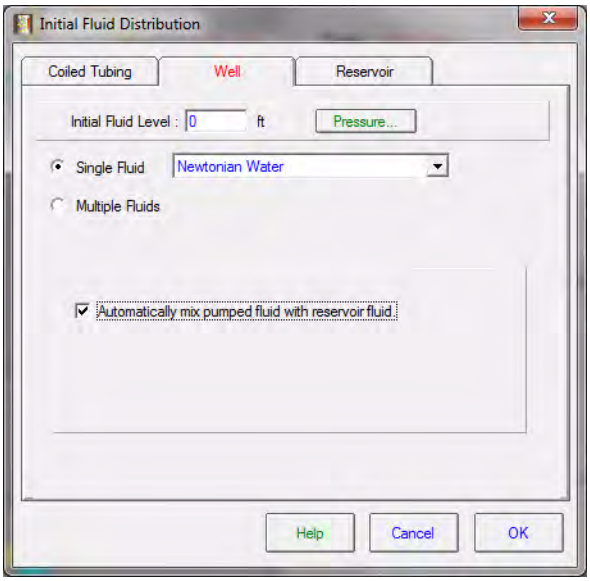
A reservoir must be defined to be included in the calculation. See Using the Well Editor for more information.

➢ To automatically mix pumped fluid with reservoir fluid

1. From the Initial Fluid Distribution display screen, select the Well tab (See "Set initial fluid distribution.")

2. Click (to toggle) the **Automatically mix pumped fluid with reservoir fluid** togglebox.

3. Click **OK**.



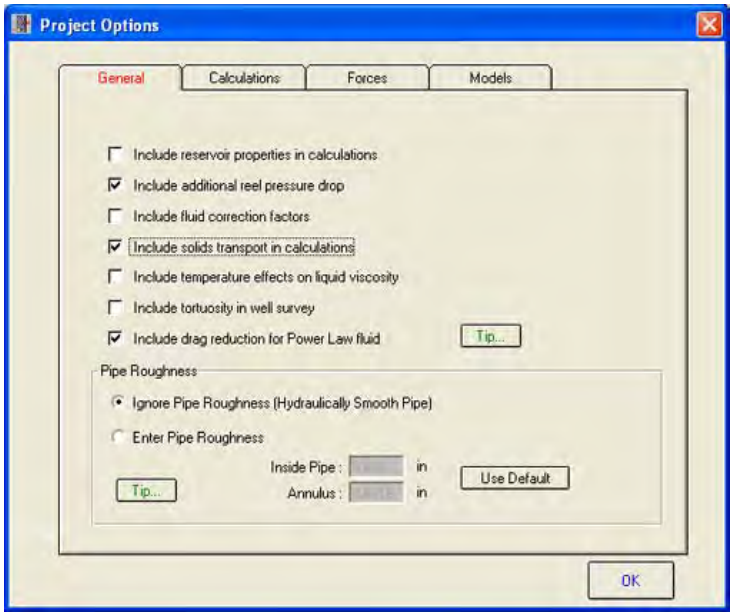
When the togglebox **Multiple Fluids** *is selected, the* **Automatically mix pumped fluid with reservoir fluid** *field is hidden.*

When the Automatically mix pumped fluid with reservoir fluid togglebox is activated, the following are disabled:
- Foam Wizard (See "Foam Wizard" on page 160.)
- Fill Removal Wizard (See "Fill Removal Wizard" on page 160.)
- Well Unloading Wizard (See "Well Unloading Wizard" on page 161.)

7. (Optional) Set advanced project options.

 ➢ To set advanced project options

 a. From the Hydra display-screen **Config** tab, click the **Project Options** button. The Project Options display screen appears.



 b. Click (to check) available toggles or input information into the four tabbed areas: **General**, **Calculations**, **Forces**, and **Models**.

Within the **Project Options > General** *tab, the item* **Include drag reduction for Power Law fluid** *is toggled "on" by default.*

As noted in the associated **Tip**, *many polymer fluids, such as guar, xanthan, PHPA, and HEC, as well as surfactant-based fluids, exhibit significant drag reduction in turbulent flow. Because the traditional Dodge-Metzner correlation overestimates the friction pressure of these fluids in turbulent flow, the effect of drag reduction should be, and is (by toggling), included in the calculation.*

8. Perform calculations using any of the methods described in Performing Calculations.

Performing Calculations

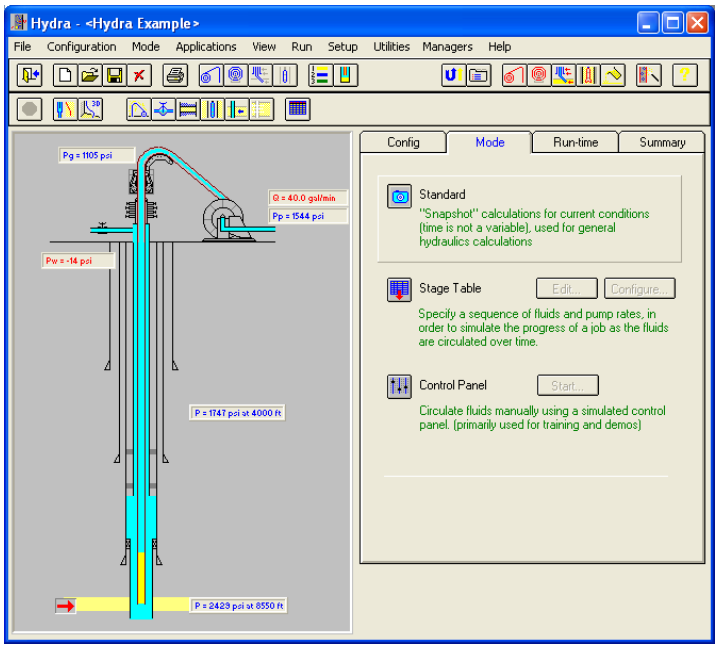
Various methods of performing calculations are available in Hydra:

- Standard "snapshot" calculations for general, current-conditions hydraulics calculations in which time is not a variable.
- Stage Table calculations, wherein a sequence of fluids and pump rates are specified in order to simulate the progress of a job as the fluids circulate over time.
- Wizard calculations, where the user is "walked through" various types of calculations, including foam, fill removal, well unloading, multiphase, analysis, and pump.

Standard “Snapshot” Calculations

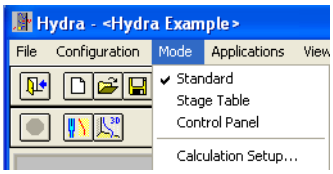
➢ To specify the Standard calculation method

Select the Mode tab from the Hydra display screen:



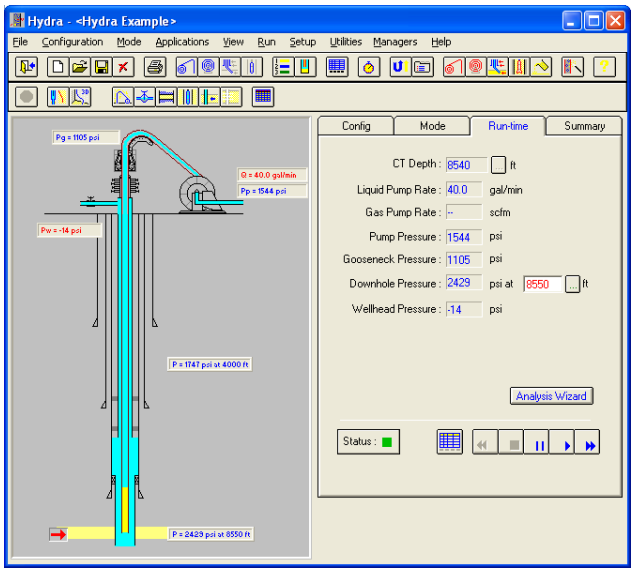
OR

Select the Mode menu-bar item from the Hydra display screen:



➢ To configure and run standard calculations

1. Select the **Run-time** tab from the Hydra display screen.



2. Edit any red-colored parameter, as desired.

3. Click the **Update** button.

4. Hydra calculates the values in blue on the **Run-time** tab and on the diagram.
 - View calculated values on the **Run-time** tab and the Summary tab.
 - Left-click the **Well View** area of the diagram to see graphs.

Advancing Fluids in a Standard Calculation

When a standard calculation is first performed, Hydra uses the initial fluid distribution. Fluids can then be advanced through the system using a method similar to the Control Panel, but with more control.



Advance fluids using the following methods:

- **Advance the fluids by volume:** Equivalent to pumping the specified volume of the pump fluid.
- **Advance next fluid interface to depth**. Equivalent to pumping the pump fluid until there is a new fluid at the specified depth in the annulus. For example, suppose there is fresh water in the annulus at 5000 ft, followed by acid. When the next fluid interface is advanced to 5000 ft, pump the pump fluid until the acid reaches 5000 ft in the annulus.
- **Advance treatment fluid**: Equivalent to pumping the pump fluid until the specified fluid reaches the specified depth in the annulus. For example, suppose there is fresh water in the annulus at 5000 ft, followed by acid, followed by gel. When the gel is advanced to 5000 ft, pump the pump fluid until the gel reaches 5000 ft in the annulus.

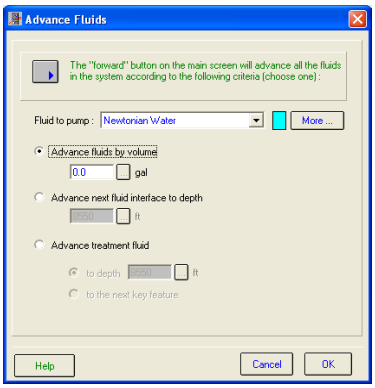
➢ To turn on the advance-fluids feature

1. Under the **Setup** menu, select **Hydra Preferences**.
2. From the **Features** tab, select **Enable Displacement Buttons**.

➢ To advance fluids in the standard calculation

1. Perform an initial standard calculation.
2. Click the … button to access the Advance Fluids screen.

Advance Fluids



- Select parameters to test. Click **OK**.

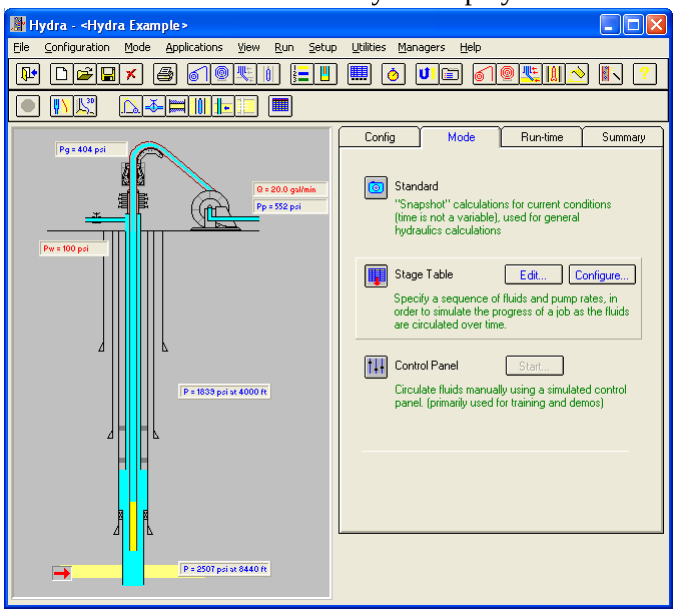
3. Click the **Forward** button on the main screen.
4. Hydra advances the fluid distribution by the amount specified and calculates the outputs for the new fluid distribution.
 - Continue to click **Forward** until the current criteria for advancing the fluids no longer makes sense.
 - Change the Advance Fluids parameters for further testing; then click **Forward**.
 - Reset to the initial fluid distribution by clicking the **Reset** button.

Stage-table Calculations

Hydraulics calculations can often be broken down into stages. A “stage” is described as a specific depth range to which a single fluid is pumped. A stage table stores flow and depth data describing a sequence of events over time. In stage-table calculations, the stage table is “played” so that Hydra steps through the sequence of events and calculates additional data for each event.

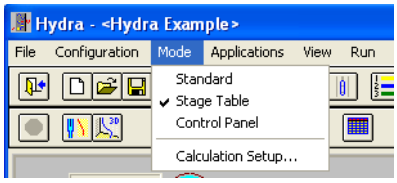
➢ To specify the use of the stage-table calculation method

Select the Mode tab from the Hydra display screen:



OR

Select the Mode menu-bar item from the Hydra display screen:



➢ To configure the available stage-table parameters

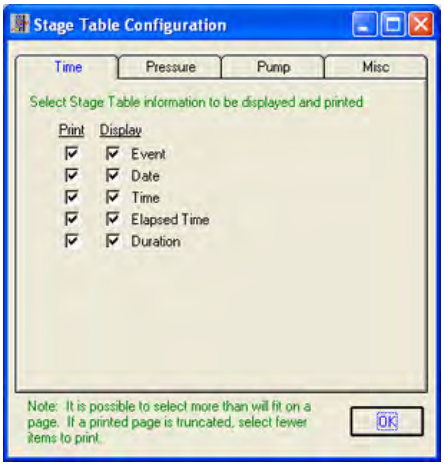
The Stage Table Configuration dialog allows the user to select Time, Pressure, Pump, and Miscellaneous parameters that dictate the availability of these parameters when displaying and printing a stage table.

1. From the **Mode tab** > **Stage Table** area in the Hydra display screen, select the **Configure** button.

 OR

 From a stage-table display screen (see To open a stage table for viewing only), select **Options** > **Stage Table Configuration** from the menu bar.

The Stage Table Configuration dialog appears.



2. Within each tab, click (to check) the various stage-table parameters to display/print.

If certain parameters are visible but not available for clicking, set Advanced Project Options to rectify the situation (see page 151).

➢ To open a stage table for viewing only

- Click the **Stage Table** button on the Hydra toolbar.

➢ To edit a stage table

- On Hydra's **Mode** tab, click the **Edit** button associated with Stage Table.

 OR

- From the Hydra menu bar, select **Mode** > **Stage Table**. The Stage Table area becomes highlighted on the **Mode** tab. Click the **Edit** button associated with Stage Table.

➢ To configure the stage table

Each row in the stage table represents one event. The shorter the duration of an event, the more detailed the results.

To enter data in a row, simply type in the table. Use the following shortcuts to help facilitate data entry/configuration.

To . . .	Do this . . .
Quickly enter multiple rows describing an entire job.	Use the Job Design wizard. (see page 158)
Quickly change the fluid in a single event.	Click the fluid name, click the dropdown, choose the new fluid.

To . . .	Do this . . .
Quickly add comments to a single event.	Click the comments field; click the drop-down; choose the new comment or insert a new one.
Add a row after the current row.	Click the **Insert** button on the toolbar .
Add a row to the end of the stage table.	Click the **New Row** button on the toolbar .
Delete a row or rows from the stage table.	Select the row or rows to delete. On the **Edit** menu, select **Cut**.
Clear all the rows in the stage table.	Click the **Clear All** button on the toolbar .
See the data for a single row in dialog box format.	Use the Data Entry wizard. (see page 158)
Set all the cells in a column after a particular row to a single value.	Enter the new value in the first cell to change. Then right-click the cell and select **Copy Down** from the menu.
Expand one or more rows with long durations into multiple rows with shorter durations.	Select the rows; then click **Edit** > **Expand**. Enter the time interval for each new row; then click **OK**.
Show or hide stage-table columns.	Click **Options > Stage Table Configuration** and select the columns to view or print.

Using Stage-table Strip Charts

To . . .	Do this . . .
View strip charts	Click the View Chart button on the toolbar .
View a different strip chart.	Select a different chart in the dropdown box in the lower left.
Resize the strip chart.	Resize the strip chart window.
Create a new strip chart that will be available to all of the projects.	In the menu bar, click **File** > **New Chart**. Select the data to include on the chart.
Print a strip chart	Select **File > Print** from the Chart menu bar **OR** click the **Print** button in the lower right-hand portion of the chart. Select the printer options (including the availability of printing to PDF), and click the **Print** button.

Using the Data Entry Wizard

Use the data entry wizard to enter a single row of data in the stage table.

➢ To enter a data row with the Data Entry wizard

1. Open the stage table for the project in Hydra.
2. Click anywhere in the row of the stage table to edit.
3. Click the **Data Entry Wizard** button  on the toolbar.

Using the Job Design Wizard

Use the Job Design wizard to enter multiple data rows into the stage table.

➢ To enter multiple data rows with the Job Design wizard

1. Click the J**ob Design Wizard** button  on the toolbar.

 OR

 Press **F9**.
2. Follow the on-screen instructions.

Running Stage-table Calculations

➢ To run stage-table calculations

1. Select the **Run-time** tab on the Hydra display screen.

 OR

 Use the Hydra Stage Table toolbar buttons.

2. To run the entire stage table, click the **Run All** button .

3. To run the next line of the stage table, click the **Run Next** button .

4. To reset the stage table to the beginning, click the **Reset** button .

5. The job is done playing when the blue stop button  turns grey.

6. View the calculation results from the **Summary** tab on the Hydra display screen.

Application-wizard Calculations

Use application wizards to calculate data of particular interest to specific applications.



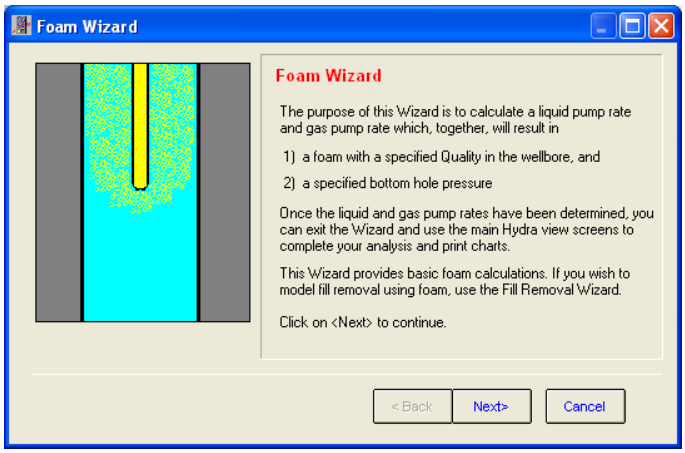
Excluding the Pump Wizard, the application wizards can be run at any time, regardless of the current calculation mode. To run the Pump Wizard, switch to the Stage Table mode.

➢ To access the application wizards

1. From the Hydra display screen's menu bar, select **Applications**.

2. Click the name of the desired wizard.

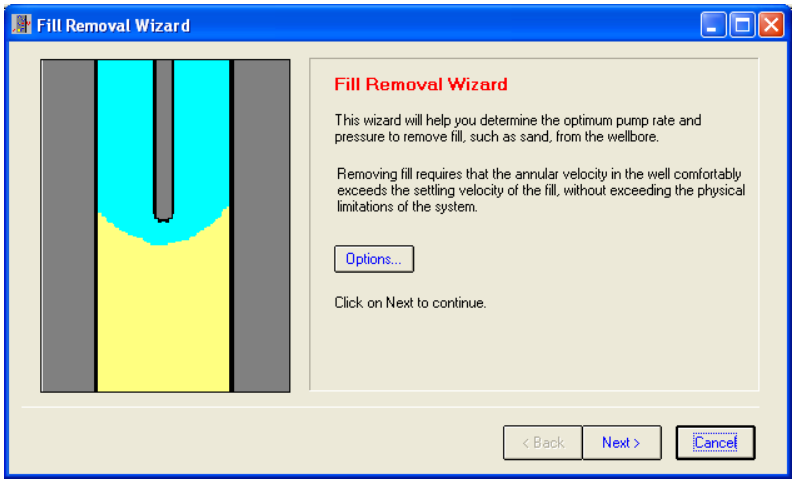
Foam Wizard



Use the Foam Wizard to optimize the fluid pump rates and pressures to achieve desired foam qualities in the wellbore. Once the liquid and gas pump rates have been determined, exit the Wizard and use the main Hydra view screens to complete the analysis and print charts. The Foam Wizard provides basic foam calculations. To model fill removal using foam, use the Fill Removal Wizard (see page 160).

Two types of foam calculations are provided in this wizard. Either a set of pump rates are calculated for the inputs or for a specified range of pump rates, the system pressures and the foam quality are found.

Fill Removal Wizard



Use the Fill Removal Wizard to calculate the minimum and maximum pump rates and pressures to remove fill, such as sand, from the wellbore. Other data of interest to a fill removal job, such as annular velocity at top of fill, lowest annular velocity, wellhead pressure, maximum pressure, bottoms up time, and maximum penetration rate are also available.

Hydra can use one of the following criteria for determining the success of a fill removal.

- The velocity factor method – based on the fact that fill is removed as long as the fluid lifts the fill particles faster than they fall. A fill removal is considered successful if the annular velocity exceeds the settling velocity by a specified amount. This method applies to vertical wells.
- The bed formation method – calculates the height of the fill material settled in the annulus. A fill removal is considered successful if the bed height is reduced to a specified amount. This method is suited

for deviated and horizontal wells. Note: In near-vertical sections (where beds do not form), this method reverts to the velocity-factor method.

The maximum pump rate is determined by the limits of the surface equipment and any limits on the downhole pressure. Many of the questions in the Fill Removal Wizard relate to finding the maximum pump rate.

If the fill removal cannot be performed as configured, the minimum pump rate to remove the fill is higher than the maximum pump rate allowed. To be able to perform the fill removal, select a lighter fluid, adjust the constraints, or experiment with other inputs.

Well Unloading Wizard

Liquid loading is a common problem in many gas wells. The well can be loaded by workover or completion fluids following a workover or completion job. Wellbores can also be loaded with produced liquid (such as water or hydrocarbon condensate in the late life of a gas well) when the reservoir pressure has decreased. In either case, the accumulated wellbore liquid needs to be removed; in other words, the well needs to be unloaded to restore the well's production.

Using coiled tubing (CT) to unload a well with nitrogen is a quick and cost-effective method to remove the liquid load of the wellbore. During a CT unloading process, CT is run into the wellbore. Nitrogen can be pumped through the CT while it is RIH or after the CT has reached a certain depth. The nitrogen aerates the liquid column in the annulus. This reduces the hydrostatic pressure of fluid in the annulus; hence, the downhole pressure is reduced. When the downhole pressure is lower than the reservoir pressure, the reservoir fluid starts to flow into the wellbore.

During a typical unloading process of a gas well, the reservoir does not produce until the bottomhole pressure is reduced below the reservoir pressure. For the upward multiphase flow in the annulus, liquid flow rate comes solely from the initial wellbore liquid. During this period, as far as the annular multiphase flow is concerned, the initial wellbore liquid behaves like a virtual liquid (water) reservoir, constantly feeding the upward annular flow with liquid. Apparently, the rate of liquid feeding the annulus flow is related to the run-in-hole (RIH) speed of the CT. In the well unloading wizard, the liquid unloading rate is calculated based on the well geometry, the CT RIH speed, and initial liquid level – as well as the target unloading depth.

Approached somewhat differently than the other Hydra wizards, the well unloading wizard is implemented on three tab views, as described below.

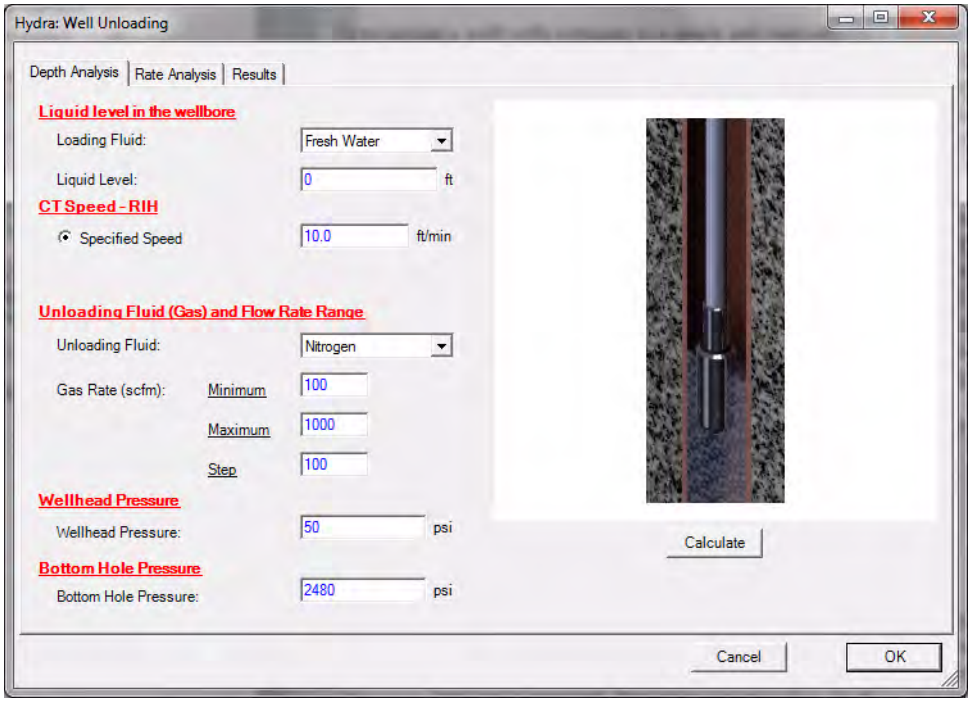


The reservoir must be defined to use this wizard.

Depth Analysis Tab

The main objective of the depth analysis is to estimate the required gas in order to achieve a given target Bottomhole Pressure if the CT is set at different depths. Typically, the deeper the CT is set, the less gas rate is required. This is because more liquid is mixed with gas so that the total hydrostatic pressure is reduced. For a given target bottomhole pressure, less gas is required. But when gas injection depth is deeper, the required pump pressure is usually greater. The result of depth analysis will be helpful for users to select a proper gas injection depth and gas rate.

Upon access, the Well Unloading Wizard's Depth Analysis tab appears similarly to the following:



1. Input salient parameters into the available fields.

 The inputs within this tab define the required input parameters for depth analysis; in other words, the analysis of required gas injection rate if the coiled tubing is set at different injection depths. More specifically:

 a. **Loading Fluid** – This specifies the liquid that has loaded the well. Only liquids are displayed in the combobox for Loading Fluid.

 b. **Liquid Level** – This is the initial liquid level of the loading liquid in the wellbore.

 c. **Specified CT RIH Speed** – This is used to estimate the liquid rate at which the well is unloaded. It is assumed that the faster the CT RIH Speed, the greater the liquid unloading rate.

 d. **Unloading Fluid** – only gases are allowed to be used as unloading fluid. This is the fluid to be pumped through the CT.

 e. **Gas Rate** – **Minimum**, **Maximum** and **Step** specify the range of gas pump rate to be simulated. The gas rate **Step** is used to break the gas rate range into a number of intervals so that the unloading hydraulics can be calculated at various gas pump rates.

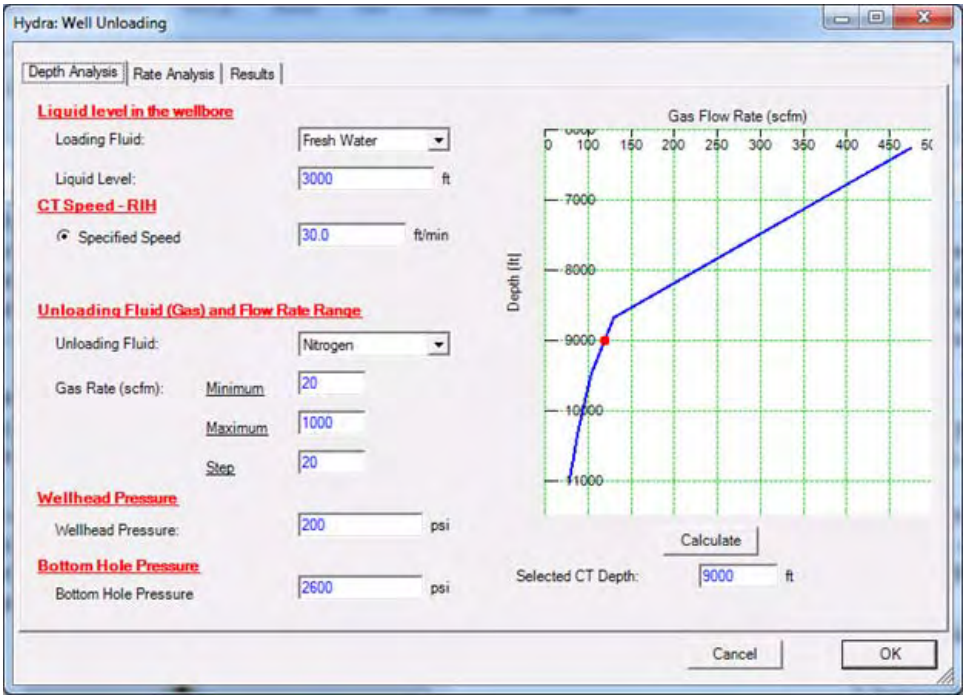
 f. **Wellhead Pressure** – this is the fluid pressure at the location of the wellhead. It is used as one boundary condition for the hydraulics calculation.

 g. **Bottom Hole Pressure** – this is the target pressure to be achieved by unloading. This Bottomhole Pressure and the Wellhead Pressure define the two boundary conditions for pressure calculations.

 h. **Calculate button** – click this button to obtain the plot of depth analysis. The depth analysis plot is displayed in the area on the right portion of the tab page.

2. Click the **Calculate** button to view a display screen similar to the following.

Well Unloading Interface, Depth Analysis Tab

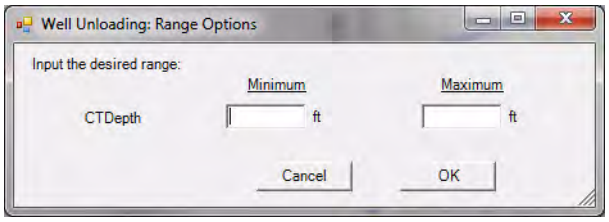


The chart plots **Depth** vs. **Gas Flow Rate** loaded from Hydra's calculations. A red dot is displayed showing the **Calculated CT Depth** if no **Selected CT Depth** is loaded from Hydra. Otherwise, the red dot shows the **Selected CT Depth**, and the graph does not represent actual data from Hydra.

a. **Selected CT Depth** – this is the CT depth the user selected for Rate Analysis. To select a **Selected CT Depth** directly on the graph (rather than inputting a value in the available field), left-click in the desired position on the graph.

3. Right-click the graph to open a context menu displaying various user-available options: **Show Range Lines, Clear Range Lines,** or **Zoom Out**. **Clear Range Lines** is initially disabled.

a. Click **Show Range Lines** to view the Well Unloading: Range Options dialog.



Input the desired range; click **OK**.

The ranges lines become visible on the graph.

➢ To remove the range lines from the graph

1. Right-click the graph to open the context menu.

2. Click the now-visible **Clear Range Lines** option.

b. Click **Zoom Out** from the graph's right-click menu or left-click on the graph and drag to zoom out on the graph.

➢ To zoom in on the graph

Left-click and drag the mouse over the graph.

Rate Analysis Tab

The purpose of this tab is to perform sensitivity analysis of gas pump rate on the bottomhole pressure for a given gas injection depth; in other words, the CT Depth. As indicated in the related figure (below), as N2 pump rate is increased, the bottomhole pressure is first reduced significantly. This occurs as a result of the nitrogen gas entering the annulus liquid, which reduces its density and therefore the hydrostatic pressure is decreased. At a certain N2 rate (about 340 scftm in the figure), the bottomhole pressure reaches the minimum value. A further increase of N2 gas rate (from 340 scfm in the figure) results in increased bottomhole pressure. This unexpected result occurs due to increased friction pressure loss at high N2 pump rates, which more than offsets the decrease in hydrostatic pressure that is achieved with the additional N2 being pumped. This result helps the user to select the optimum gas pump rate.

1. Input salient parameters into the available fields. A few inputs are different from those in the **Depth Analysis** tab:

 a. **Effective Unloading Time** – the time it takes for CT to reach from the initial Liquid Level to the target CT Depth. This can be calculated based on the CT RIH Speed. Or, if the Effective Unloading Time is given, the CT RIH Speed can be calculated.

 b. **CT Depth** – the target unloading depth.



If coiled tubing depth is not loaded from Hydra and the user does not click the **Calculate** *button on the Depth Analysis tab, the* **CT Depth** *field defaults to -1.*

If the coiled tubing depth is loaded from Hydra or the user clicks the **Calculate** *button on the Depth Analysis tab, the value for* **CT Depth** *is the* **Selected CT Depth**.

 c. **Plot of Bottomhole Pressure vs. Gas Flow Rate** – gives the graphical result of the effect of gas pump rate on the bottomhole pressure.

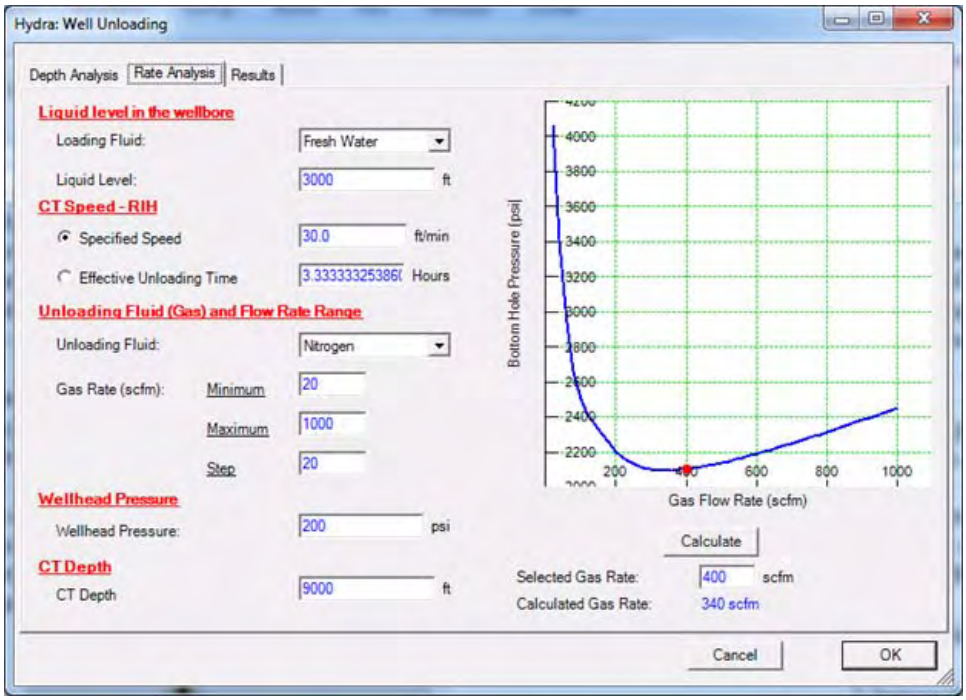
 d. **Selected Gas Rate** – the user-selected gas rate, based on the result shown on the plot. This value is used in the result calculation. To change the selected gas rate, left-click the graph in the desired area, or input the rate directly into the field.



If no data was loaded from Hydra for selected gas rate, the **Selected Gas Rate** *field is hidden.*

2. Click the **Calculate** button to view a display screen similar to the following.

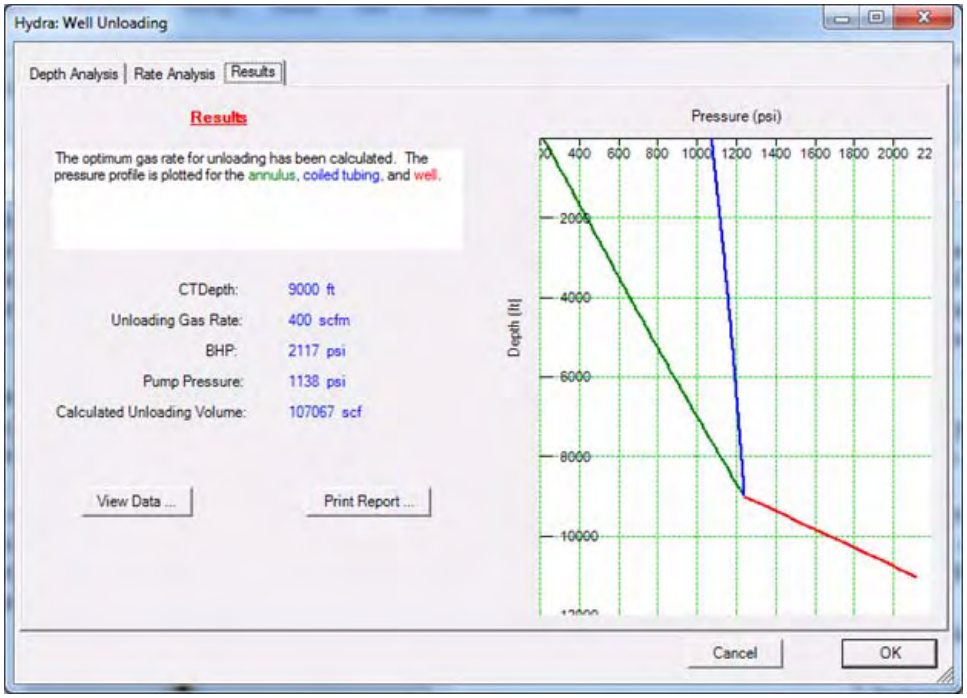
Well Unloading Interface, Rate Analysis Tab



Results Tab

This tab summarizes the results of the unloading wizard analysis, providing detailed hydraulics results for a given gas injection depth and given gas pump rate. A brief summary includes **CT Depth**, **Unloading Gas Rate**, **BHP**, **Pump Pressure**, and total required gas volume (**Calculated Unloading Volume**). The pressure profiles in the CT, annulus and well sections are displayed on the plot of **Pressures vs. Depth**, with different line colors (blue, green and red) representing different flow paths. The **View Data** button provides the option to display the detailed hydraulics data in tabular format. The **Print Report** button allows the user to print wizard results.

Well Unloading Interface, Results Tab

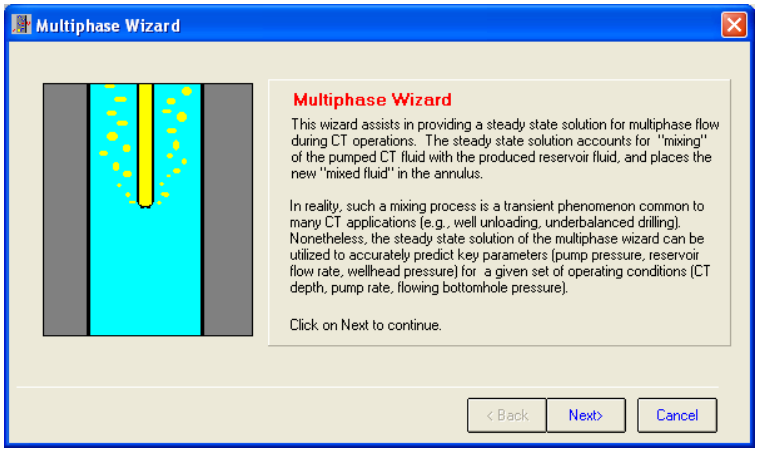


- Click **Zoom Out** from the graph's right-click menu or left-click on the graph and drag to zoom out on the graph.

➢ To zoom in on the graph

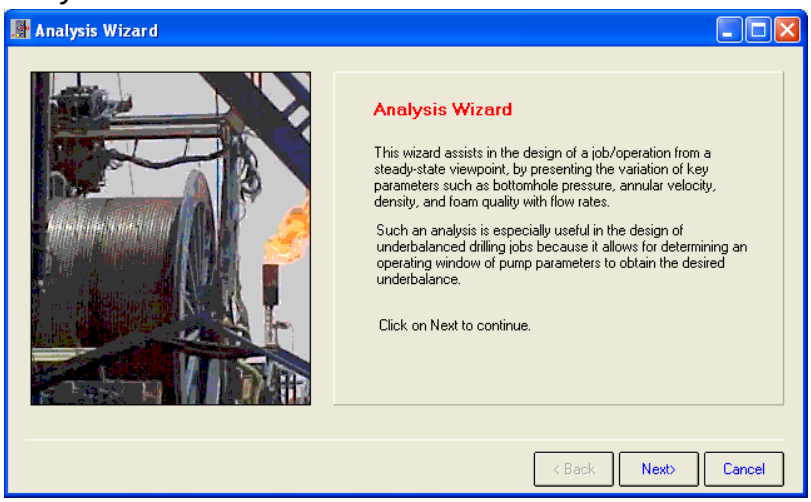
Left-click and drag the mouse over the graph.

Multiphase Wizard



Use the Multiphase Wizard to calculate a steady state solution for a multiphase fluid.

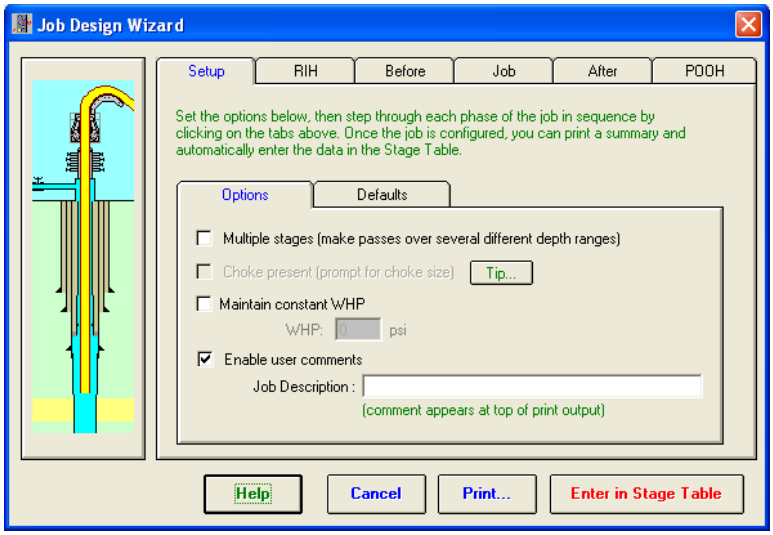
Analysis Wizard



The Analysis Wizard is used to see how varying the pump rate affects key parameters (such as the bottom-hole pressure, annular velocity, fluid density, and foam quality).

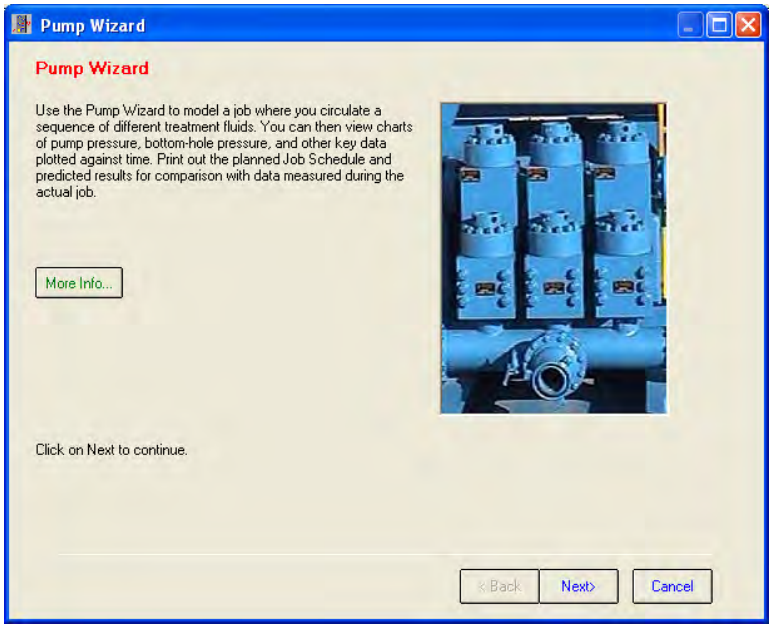
If a multiphase fluid was selected, the wizard analyzes each combination of liquid and gas pump rates.

Job Design Wizard



The Job Design Wizard is used to enter multiple rows into the stage table (see page 154).

Pump Wizard



The Pump Wizard is used to model a stage table job where a sequence of different treatment fluids can circulate. Charts of pump pressure, bottom-hole pressure, and other key data plotted against time are viewable. Inputs for this wizard are CT depth, various treatment fluids used in the displacement process, fluid volumes, and pump rates.

Chapter 7
Using Velocity String

This section describes the process involved in using Velocity String, the Cerberus Hydra-based model for coiled tubing and jointed pipe gas wells.

A velocity string is a small-diameter tubing string run inside the production tubing to resolve the problem of insufficient velocity available to transport liquids from the wellbore. These liquids may, in time, accumulate and impair production. Installing a velocity string reduces the flow area, thereby increasing the flow velocity, which enables liquids to be carried from the wellbore.

The Velocity String model calculates the best characteristics of an intended velocity string to optimize its effectiveness in an intended application.

The Velocity String Process

The process involved in using the Velocity String model is as follows:

1. Create a new Velocity String project or select an existing project (see page 171).



Velocity String (along with the majority of other Cerberus models) is organized by projects.

2. Select a well with the reservoir defined (see page 175).
3. Verify reservoir parameters (see page 176).
4. Select one or more velocity string(s) (see page 177).
5. Calculate (see page 179).
6. View results and print (see page 180).

Accessing Velocity String (Hydra)

➢ To access the Velocity String model from the Cerberus home page

- Select the Velocity String button.

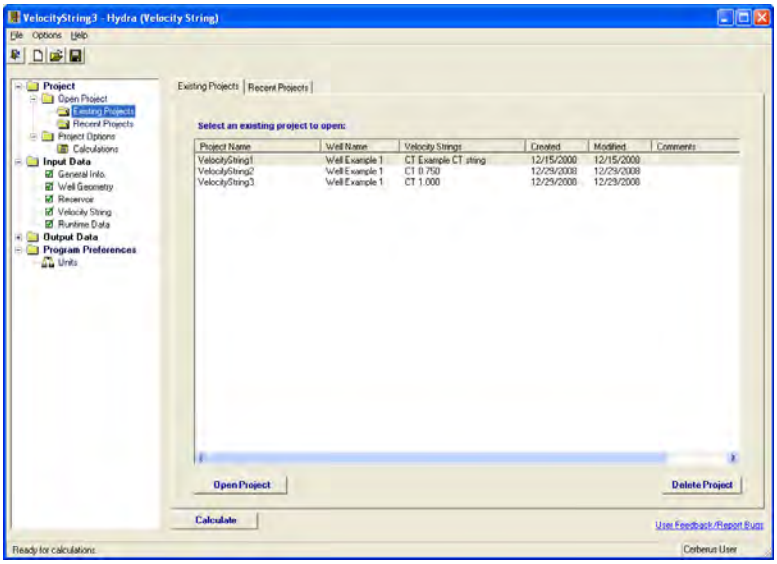


OR

- Select **Models** > **Velocity String** from the menu bar.

 The Velocity String model loads, and the opening dialog of the Hydra (Velocity String) model appears.

Velocity String Opening Dialog



Working with Projects

The Velocity String model is organized by projects.

Using the Velocity String Opening-dialog Tools

The Velocity String opening-dialog toolbar contains the following tools:

Velocity String Main Toolbar

Tool Button	Tool Name	Function
	Exit	Exits Velocity String.
	New	Opens the New Velocity String Project dialog to create a new project.
	Open	Opens the Select Job dialog box to open an existing job.
	Save	Saves the current project to disk.

Setting Program Preferences

➢ To set program preferences

- Select **Options > Units** from the menu bar.

 OR

- Click + to expand the tree's **Program Preferences** node; click **Units**.

Creating a New Velocity String Project

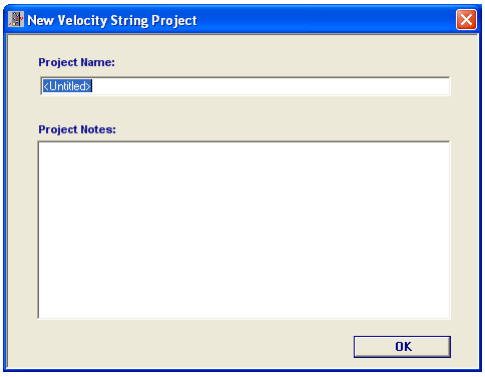
The following procedure outlines how to create a new Velocity String project.

➢ To start a new project

1. From the Velocity String Opening Dialog, choose one of the following:
 - Select **File > New** from the menu bar
 - Click the **New** toolbar button: 

 The **New Project** dialog appears:

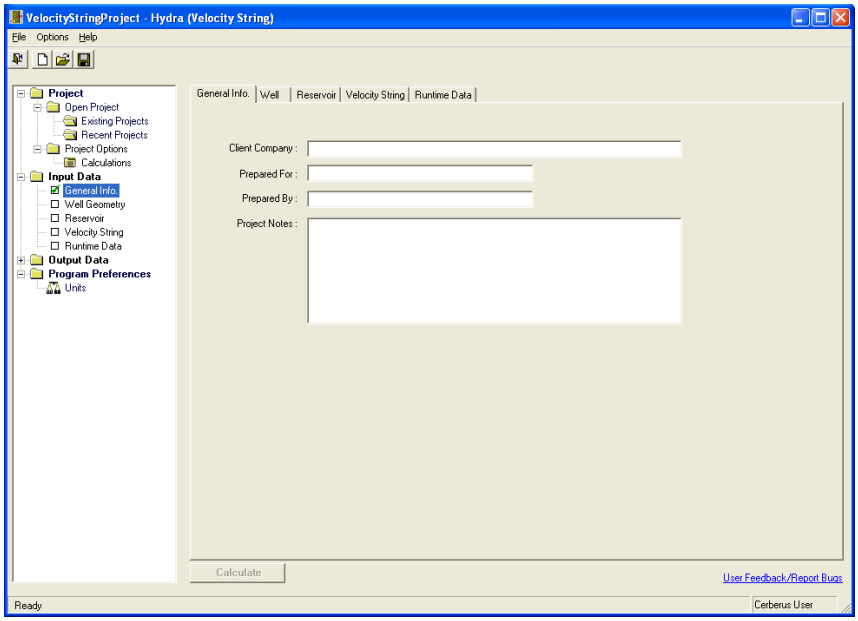
New Velocity String Project dialog



Start a new Velocity String project directly from the Cerberus home page by selecting **File** > **New** > **Velocity String Project**.

2. Provide a new name for the project. The name may consist of letters, numbers, and spaces, with a maximum of 32 characters.

 The Velocity String configuration display-screen appears.



3. Configure the project (see Entering Project-configuration Information for more information).

Opening an Existing Velocity String Project

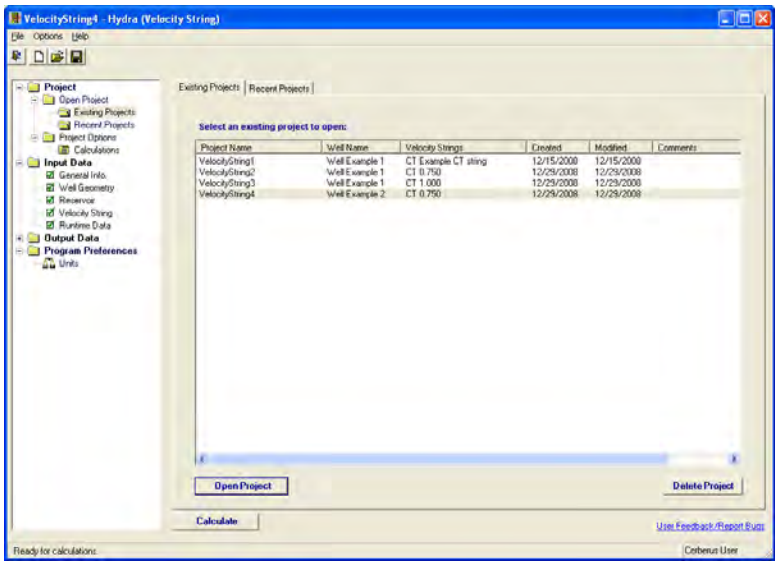
The following procedure describes how to open an existing project.

➢ To open an existing project

1. Choose one of the following from the Velocity String opening dialog:
 - Select **File** > **Open** from the menu bar
 - Click the **Open Project** toolbar button: 

- From the tree on the left-hand portion of the display screen, select **Existing Projects** or **Recent Projects** within the **Open Project** folder.

2. The following Hydra (Velocity String) display screen appears, opened to the (default) Existing Projects tab.



3. Select the desired project within the tabpage on the right-hand portion of the display screen, and click the **Open Project** button in the tabpage.

From the Cerberus main menu, double-click the project name in the left-hand tree to open an existing project.

Saving a Velocity String Project

The project configuration and last set of calculated values is retained when the project is saved. There are two ways a project can be saved:

- Click **Save** () on the toolbar.

 OR

- On the menu bar, click **File** > **Save** or **Save As...**.

Deleting a Velocity String Project

When a project is deleted, the information is not recoverable.

➢ To Delete a Project

- From the menu bar, select **File** > **Delete**.
- From the Velocity String Opening Dialog, select the project and click the **Delete Project** button.

Importing a Velocity String Project

Import any project that is on removable media into Velocity String.

➢ To Import a Project

1. If importing from removable media, insert the media in the drive.
2. Access the Velocity String model.
3. From the menu bar on the Velocity String Opening Dialog, click **File** > **Import...**.

Exporting a Velocity String Project

Export any project in Velocity String to removable media for importing to another computer with Velocity String.

➢ To Export a Project

1. Access the Velocity String model.
2. From the menu bar on the Velocity String Opening Dialog, click **File** > **Export...**.

Entering Project-configuration Information

Configuration information in the Velocity String model is accessible/editable via The Hydra (Velocity String) display screen. View/edit information by clicking the associated tab, or by selecting the related item in the tree.



Configuration information is stored within a project, so it is quickly accessible without the necessity of re-input.

Select a Well with a Defined Reservoir

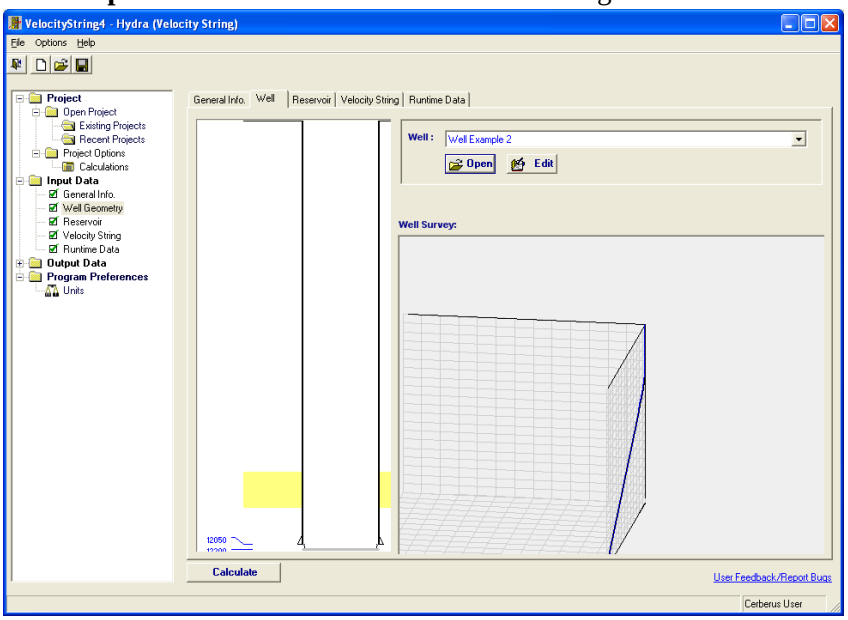
➢ To select a well

1. Within the Hydra (Velocity String) display screen:
 - Click the **Well** tab.

 OR
 - Select **Well Geometry** from the tree.
2. Use the **Well** dropdown to select an existing well.

 OR

Click the associated **Open** button to access the Select Well dialog.



3. (Optional) To edit the selected well, click the associated **Edit** button. The Well Editor appears.

Verify Reservoir Parameters

The reservoir model is used to calculate reservoir IPR curves. When a project is opened in the Velocity String Model, the reservoir parameters default to those specified in Well Manager. The parameters for the current session can be changed in Reservoir tab of the Hydra (Velocity String) display screen. However, the next time the project is opened, it will once again have the original inputs specified in Well Manager.

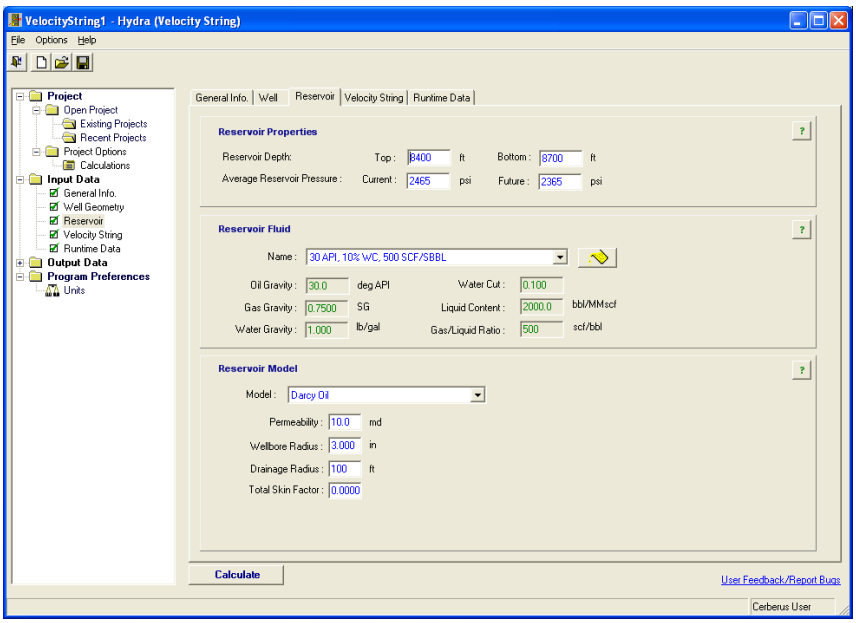
➢ To verify reservoir parameters

1. Within the Hydra (Velocity String) display screen:
 - Click the **Reservoir** tab.

 OR

 - Select **Reservoir** from the tree.

2. Ensure that the reservoir parameter-fields are filled as desired.



Current reservoir pressure is the average reservoir pressure currently in the drainage volume. Future reservoir pressure is the average reservoir pressure at a future depleted reservoir condition.

Only a multiphase fluid can be selected as a reservoir fluid. If this type of fluid does not currently exist in the software, it can be created in Fluid Manager .

If the reservoir fluid is a wet gas, then Darcy Gas and Fetkovich models can be used. If the reservoir fluid is not gas in the reservoir, then Constant Productivity Index, Darcy Oil, Fetkovich, and Vogel-Standing models are allowed.

Select One or More Velocity Strings

➢ To select velocity string(s)

1. Within the Hydra (Velocity String) display screen:
 - Click the **Velocity String** tab.

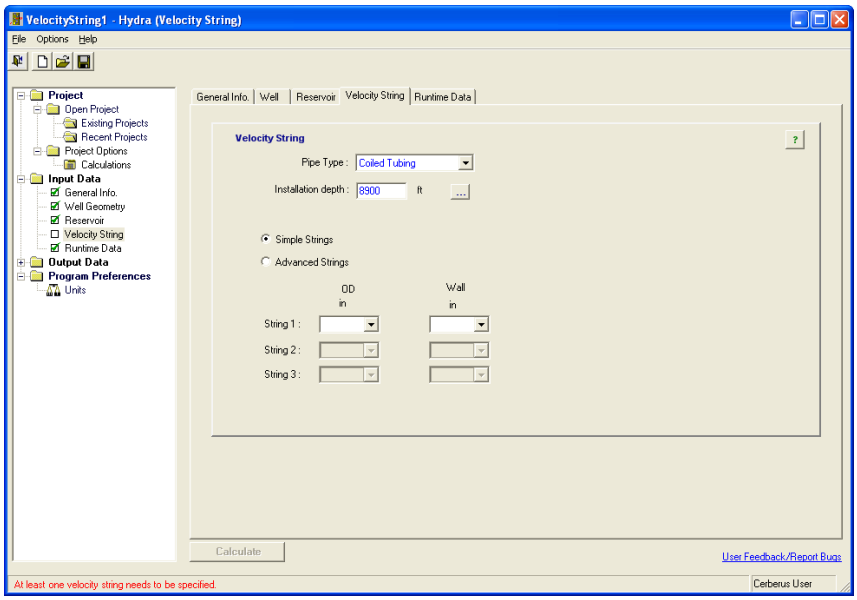
 OR

 - Select **Velocity String** from the tree.
2. Select the pipe type and installation depth.
3. Select between Simple and Advanced string types.

 A Simple String refers to tubing string that has only one OD and wall thickness while an Advanced String can consist of tubing sections with different tubing wall thicknesses.

a. If Simple Strings is toggled:

Use the associated dropdowns to select tubing OD and Wall sizes.

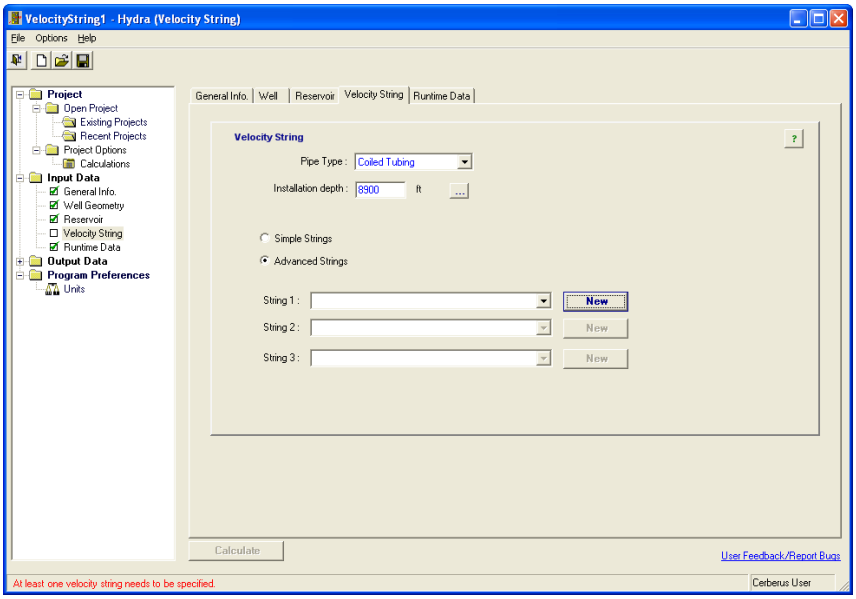


b. If Advanced Strings is toggled:

Use the associated dropdowns to select existing string(s).

OR

Click the **New** button to access the Select String dialog, from which the user may select an existing string or create a new string.

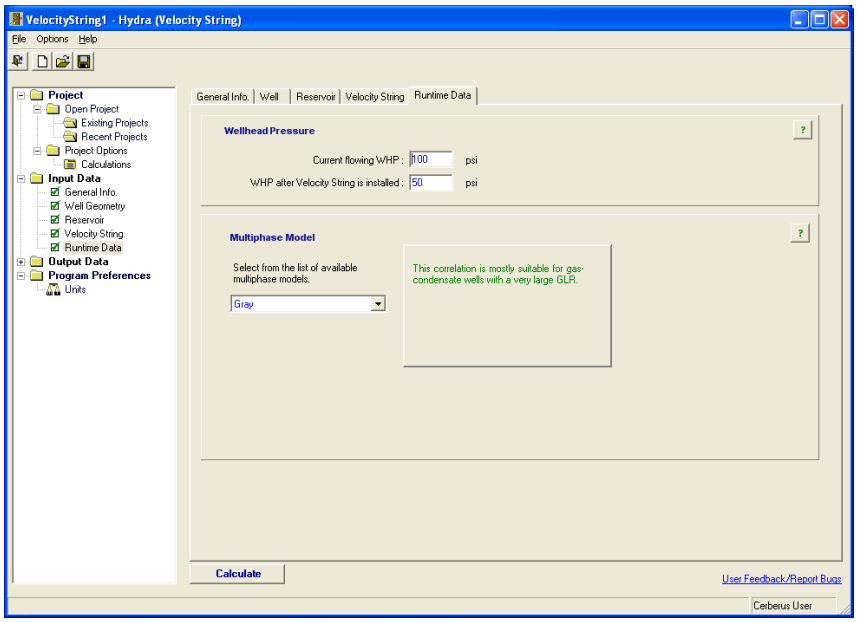


Enter Runtime Data

➢ To enter runtime data

1. Within the Hydra (Velocity String) display screen:
 - Click the **Runtime Data** tab.

 OR
 - Select **Runtime Data** from the tree.



2. Ensure the Wellhead Pressure and Multiphase Model information is in place as desired. See Verify Reservoir Parameters for more information about multiphase fluids.

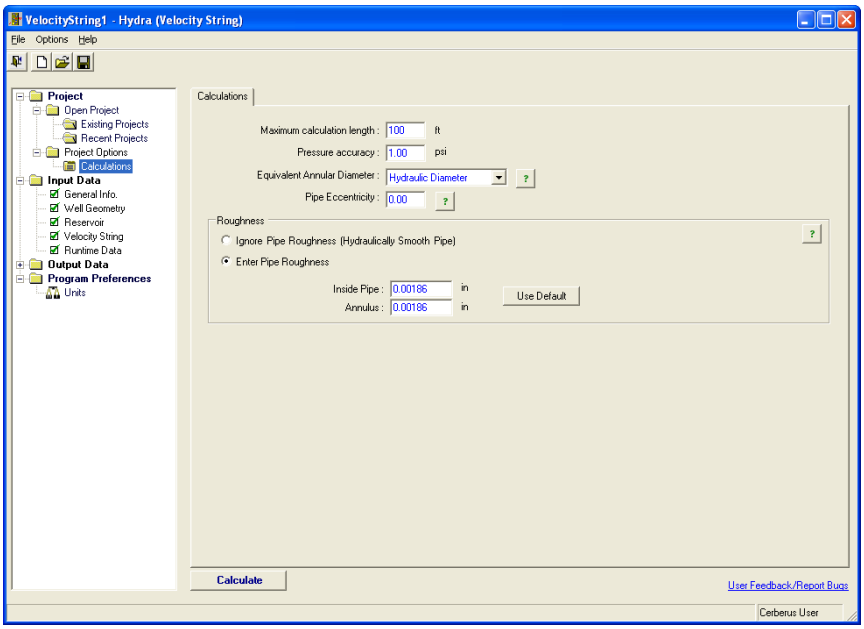
Calculating Data

The user may customize the manner in which the Velocity String model calculates data.

➢ To customize calculation options

1. Create or open the project.
2. Click the **Calculations** button Calculations under the Project Options folder in the menu tree.
3. Perform edits as needed.

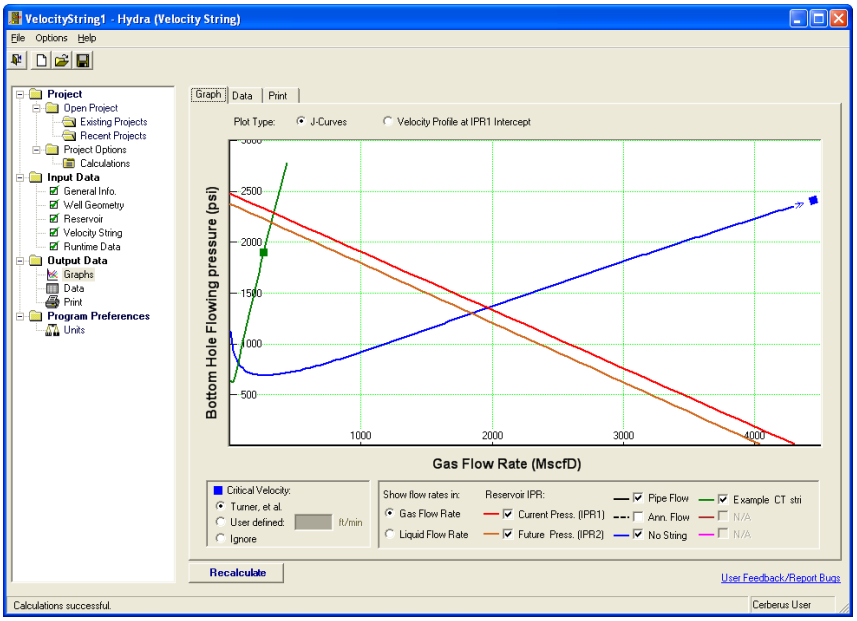
4. Select the type of friction calculations to use (smooth pipe or rough pipe). If rough pipe is selected, enter the roughness for the inside pipe and annulus. Roughness has no effect on flows in the laminar regime.



➢ To perform the calculation

1. After all parameters are in place as desired, click **Calculate** from any of the Velocity String tabs.

2. If all parameters contribute to a successful calculation, the calculation output becomes available, as follows:



Using the Output Data

➢ To view the velocity string graph (default)

Click the **Graph** tab

OR

Select **Output Data** > **Graphs** from the left-hand tree.

➢ To view the velocity string data

Click the **Data** tab

OR

Select **Output Data** > **Data** from the left-hand tree.

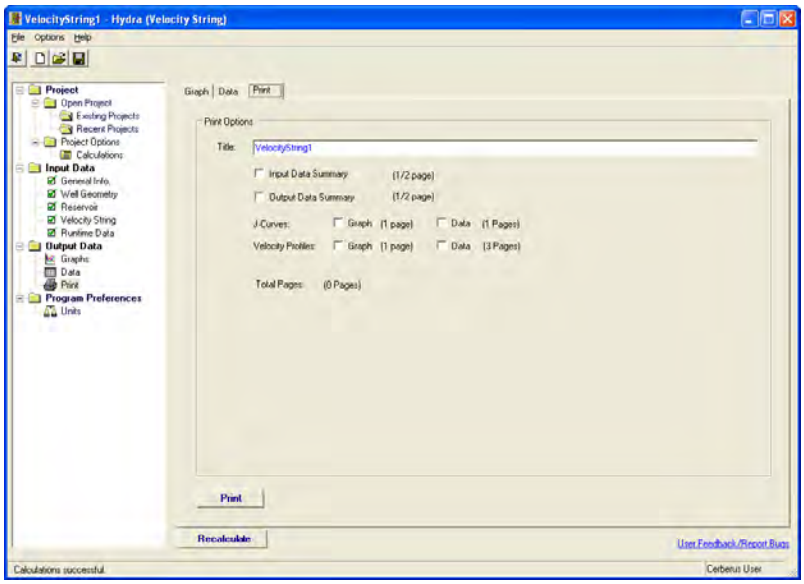
➢ To print the velocity string data

1. Click the **Print** tab

 OR

 Select **Output Data** > **Print** from the left-hand tree.

2. Toggle the options desired.



3. Click the **Print** button.

Chapter 8
Using the Packer and Completion Analysis (PACA)

This section describes the process involved in using PACA, the Cerberus model/wizard that calculates what happens to a packer set on the end of production or workover tubing if conditions change at surface or down-hole.

About PACA

Change in conditions at surface or downhole may cause a packer set on the end of production or workover tubing to experience movement. Buckling in the tubing may also occur as an effect of condition change. The resulting forces on the packer or tubing may be damaging.

“Change in conditions” may include any combination of:

- Set-down or pickup of the tubing at surface.
- Change of temperature-profile downhole.
- Change of fluids in the tubing and/or annulus.
- Change in the fluid-flow rate.
- Change of pressure on the tubing or annulus.

The Packer and Completion Analysis wizard (given the change of conditions) calculates:

- Tubing movement at the packer, if any (depending upon packer type).
- Force acting on the packer communicated by the tubing.
- Force on the casing communicated by the packer.

The PACA Process

The process involved in using PACA is as follows:

1. Create a new PACA project or select an existing project (see page 186).



PACA (along with the majority of other Cerberus models) is organized by projects.

2. Select the application (see page 189). Depending on the application-type chosen, the wizard follows one of two paths:

 a. Permanent tubing completion

 b. Temporary packer run on coiled tubing or pipe

3. Select a well (see page 191). For permanent tubing completion, select a well with a tubing string and a packer (see page 190).

4. For a temporary packer run on CT or pipe, select the CT or pipe (see page 192).

5. For a temporary packer run on CT or pipe, select a tool with a packer (see page 192).

6. Configure the packer properties (see page 194).

7. Define initial conditions (see page 195).

8. Define one or more subsequent scenarios (see page 195).

9. Analyze; view the analysis and print a report (see page 196).

Accessing the PACA Model/Wizard

➢ To access PACA from the Cerberus home page

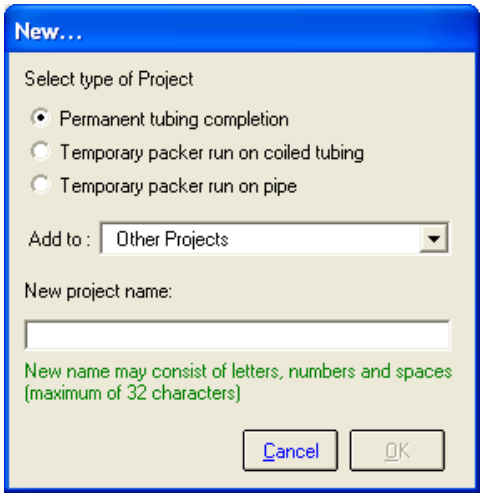
- Select the PACA button.



OR

- Select **Models** > **PACA** from the menu bar.

 If a PACA project is not already open, the New... dialog appears.



Setting Program Preferences

➢ To set program preferences

Select **Options** > **Units** from the menu bar.

Accessing the Well Manager and Fluids Manager

➢ To access the Well Manager or Fluids Manager

Select, from the menu bar, **Managers** > **Well Manager**, or **Managers** > **Fluids Manager.**

Working with Projects

The PACA model/wizard is organized by projects.

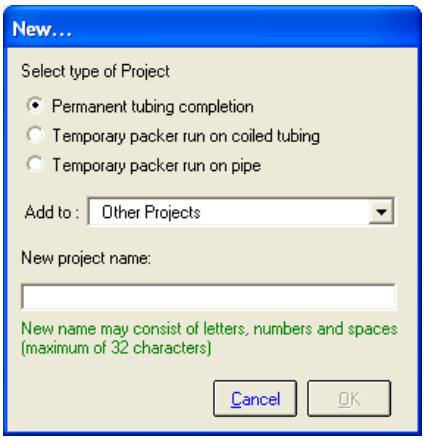
Creating a New PACA Project

The following procedure outlines how to create a new PACA project.

➢ To start a new project

1. Choose from the following:
 - From the Select Project dialog, click the **New** button.
 - Select **File > New** from the Packer and Completion Analysis menu bar.

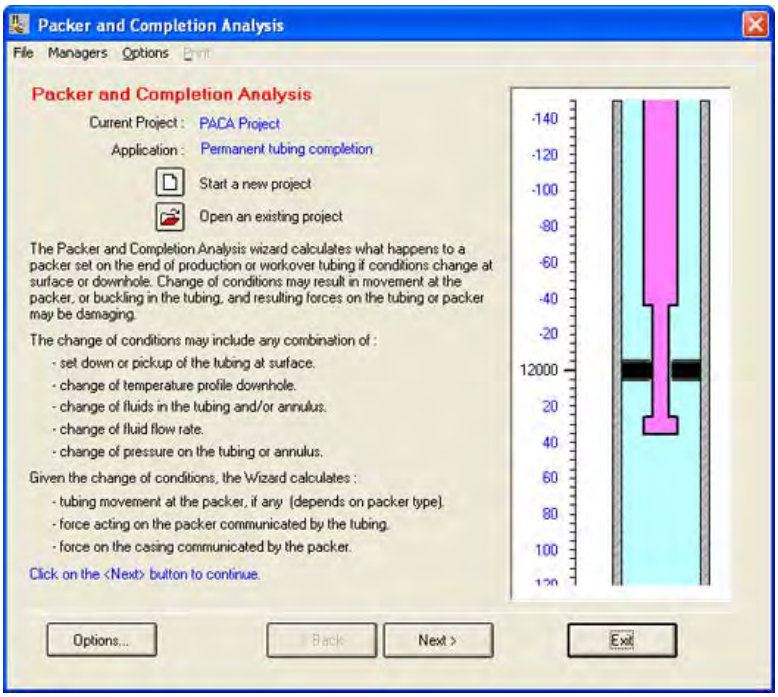
 The **New...** dialog appears:



Start a new PACA project directly from the Cerberus home page by selecting **File > New > PACA Project**.

2. Provide a new name for the project. The name may consist of letters, numbers, and spaces, with a maximum of 32 characters.

 The Packer and Completion Analysis wizard display screen appears.



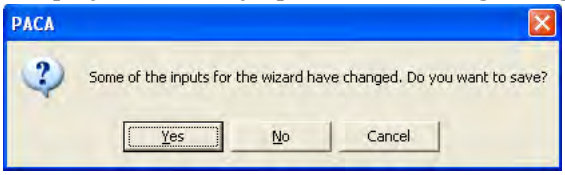
Opening an Existing PACA Project

The following procedure describes how to open an existing project.

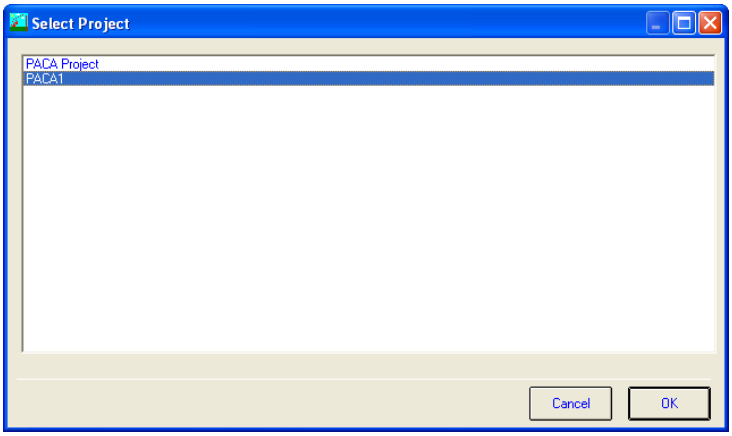
➢ To open an existing project

1. Select **File > Open** from the menu bar.

2. If a project is already open, the following dialog appears:



3. After choosing whether or not to save an open project, or if no project was previously open, the following Select Project dialog appears:



4. Select the desired project, and click **OK**.

 The Packer and Completion Analysis wizard display screen appears.

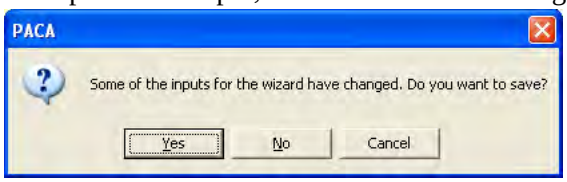
Saving a PACA Project

➢ To save a project

- On the menu bar, select **File** > **Save** or **Save As...**.

 OR

- After parameter input, click **Exit**. The following dialog appears:



Deleting a PACA Project

When a project is deleted, the information is not recoverable.

➢ To delete a project

- From the Packer and Completion Analysis menu bar, select **File** > **Delete**.

Importing a PACA Project

Import any project that is on removable media into PACA.

➢ To import a project

1. If importing from removable media, insert the media in the drive.
2. From the Packer and Completion Analysis menu bar, click **File** > **Import**.

Exporting a PACA Project

Export any project in PACA to removable media for importing to another computer with PACA.

➢ To export a project

- From the Packer and Completion Analysis menu bar, click **File** > **Export**.

Using the PACA Wizard

The Packer and Completion Analysis wizard “walks” the user through complex configuration and design tasks, stepping through key decisions in a logical sequence, and presenting choices based on context and previous selections.

Use the **Next**, **Back**, and **Exit** buttons to move through the wizard.

Entering Calculation Options

➢ To enter Safety Factors, Calculation Options, and Hydraulic Options for a project

- Click the **Options** button 

on the main Packer and Completion Analysis display screen.

Selecting the PACA Application

➢ To select the PACA application

From the Packer and Completion Analysis display screen, select from the following application types:

- Permanent tubing completion
- Temporary packer run on coiled tubing
- Temporary packer run on pipe

Depending on the application-type chosen, the wizard follows one of two paths:

a. Permanent tubing completion

b. Temporary packer run on coiled tubing or pipe

After selecting a "temporary packer run on coiled tubing or pipe" application, go to Following the "Temporary Packer Run on Coiled Tubing or Pipe" Path.

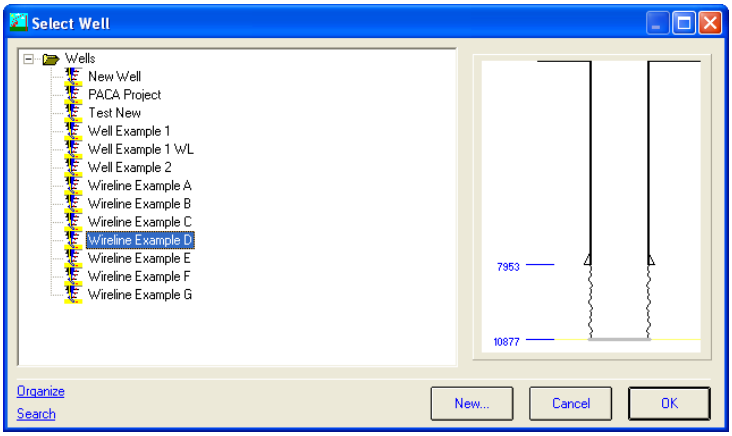
Following the "Permanent Tubing Completion" Path

After selecting a permanent-tubing-completion application, complete the following steps, as laid out in the wizard.

Selecting a Well that Includes Tubing String and Packer

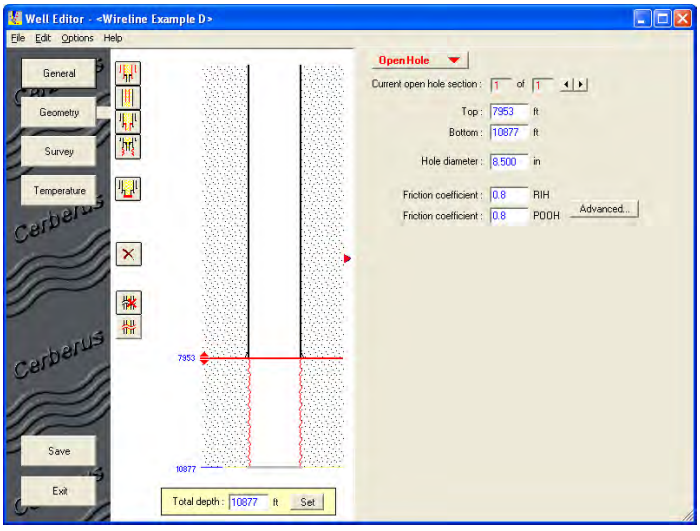
➢ To select a well that includes tubing string and packer

1. Click the yellow-folder icon beside the **Well** field to select an existing well or to create a new well.



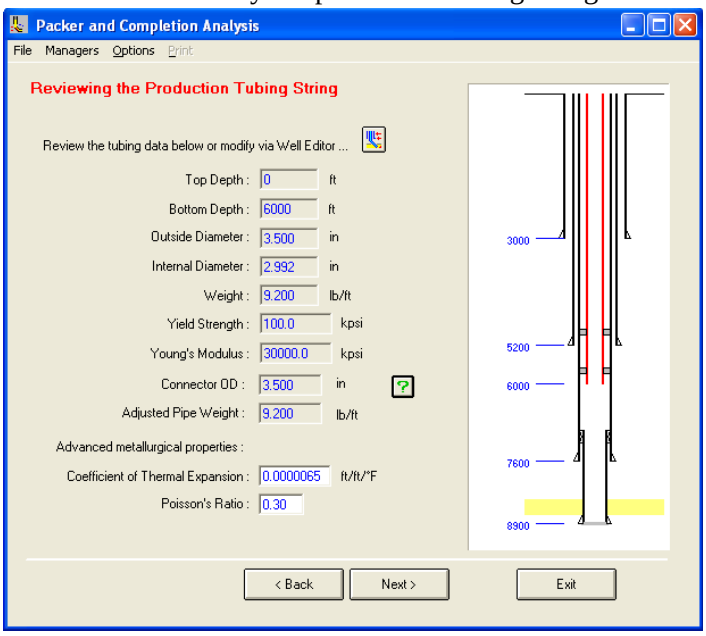
OR

Click the **Edit Well** icon (also beside the **Well** field) to edit the well shown in the field or to create a new well.



If the selected well has multiple packers, only the bottom packer is used in the calculations.

2. Review and/or modify the production tubing string.



At this point, go to Following the Converged Path to continue the permanent-tubing-completion process.

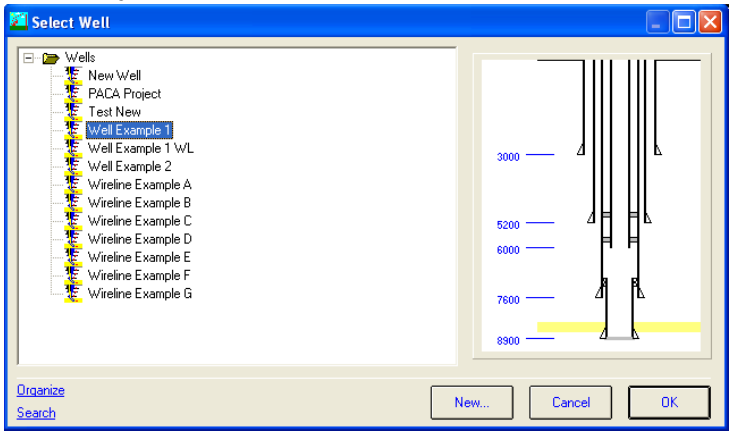
Following the "Temporary Packer Run on Coiled Tubing or Pipe" Path

After selecting a "temporary packer run on coiled tubing or pipe" application, complete the following steps, as laid out in the wizard.

Selecting a Well

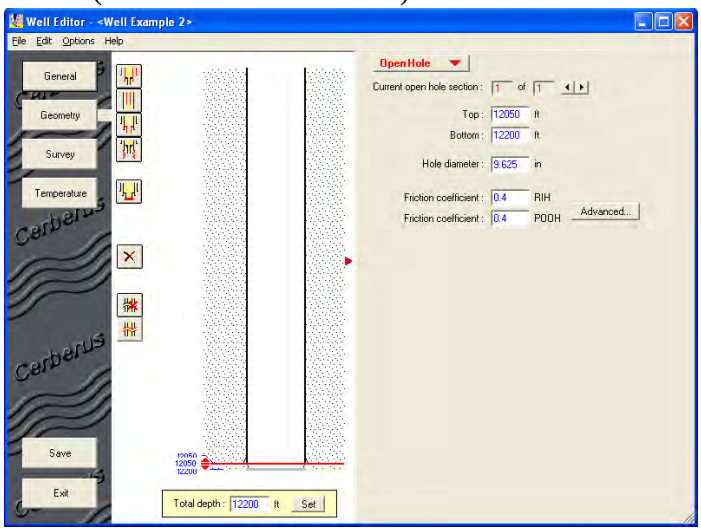
➢ To select a well

Click the yellow-folder beside the **Well** field to select an existing well or to create a new well.



OR

Click the **Edit Well** (also beside the **Well** field) to edit the well shown in the field or to create a new well.



Selecting the Coiled Tubing String or Jointed Pipe

➢ To select the CT string or jointed pipe

1. From the Reviewing... PACA-wizard display screen, click the yellow-folder icon beside the **String** field to select an existing string or to create a new string.

 OR

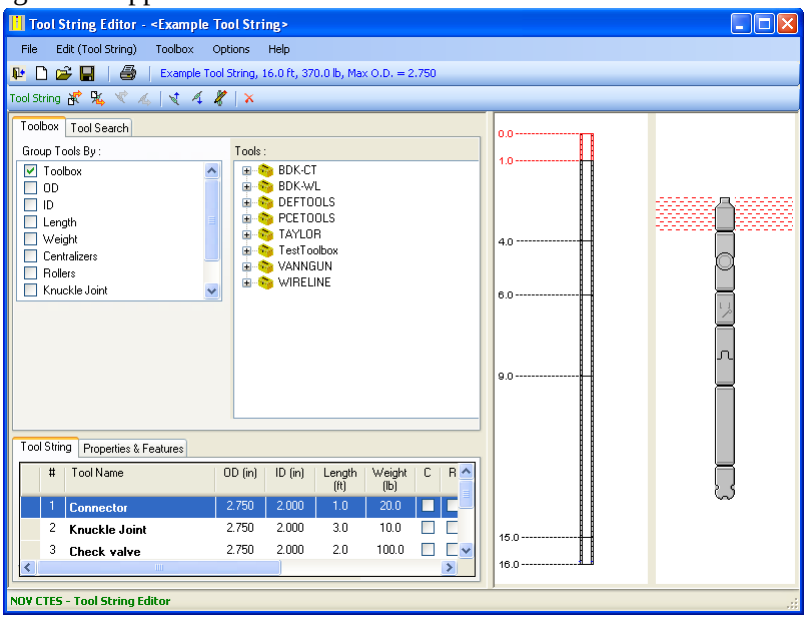
 - (Jointed Pipe) Click the Edit Pipe icon beside the **String** field to edit an existing JP workstring or to create a new string.
 - (Coiled Tubing) Click the Edit String icon beside the **String** field to edit an existing CT string or to create a new string.

Selecting a Tool with a Packer

➢ To ensure that a string includes a tool with a packer

1. From the Cerberus main menu, select, from the menu bar, **Managers > Tool String Editor**.
2. From the Select Tool String dialog, double-click the tool string intended for use in the PACA wizard.

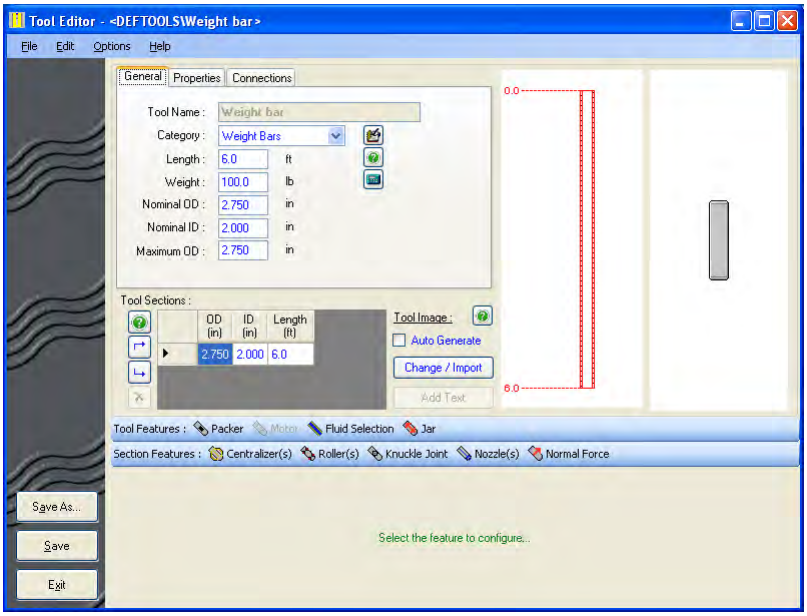
The Tool String Editor appears.



3. Within the Toolbox tab, double-click a toolbox (or click the “plus sign” (+) associated with the toolbox) to display its included tools.

4. Double-click the tool in which to include a packer.

 The Tool Editor display screen appears.



The Tool Editor is only accessible from the Tool String Editor.

5. Within the Tool Features section, double-click **Packer** Packer.

 The Packer togglebox appears.



6. Click within the togglebox and click the **Save** button (or select **File > Save** from the menu bar).

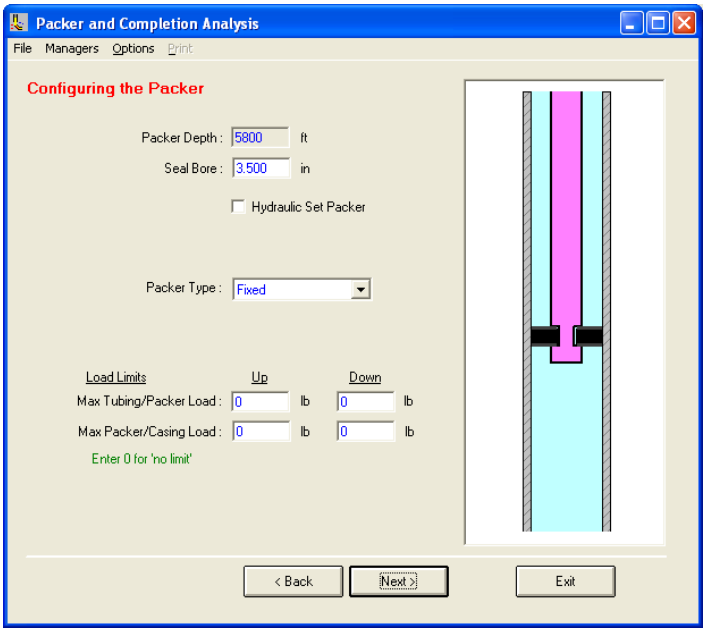
7. Close the Editors and return to the PACA wizard.

Following the Converged Path

The two paths, “permanent tubing completion” and “temporary packer run on coiled tubing or pipe,” converge at the point of configuring packer properties.

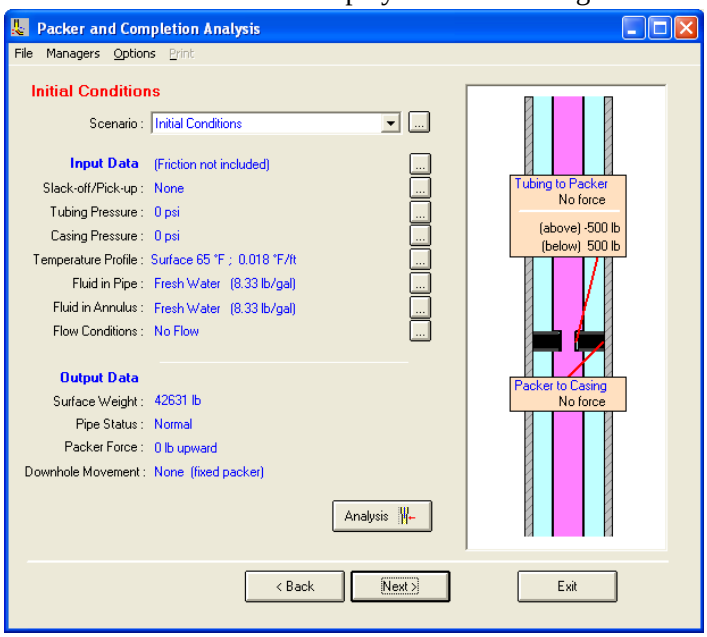
Configuring Packer Properties

Use the Configure the Packer display screen to configure packer properties.



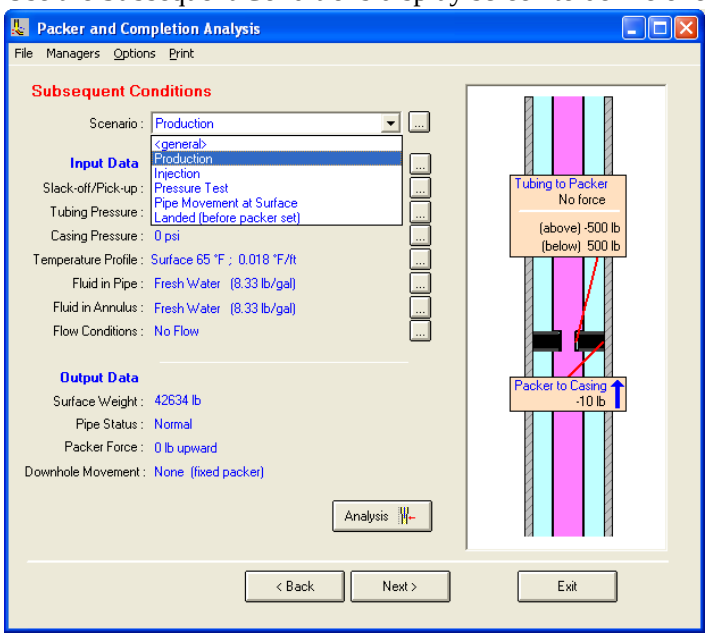
Defining Initial Conditions

Use the Initial Conditions display screen to configure initial conditions.



Defining One or More Subsequent Scenarios

Use the Subsequent Conditions display screen to define one or more subsequent scenarios.



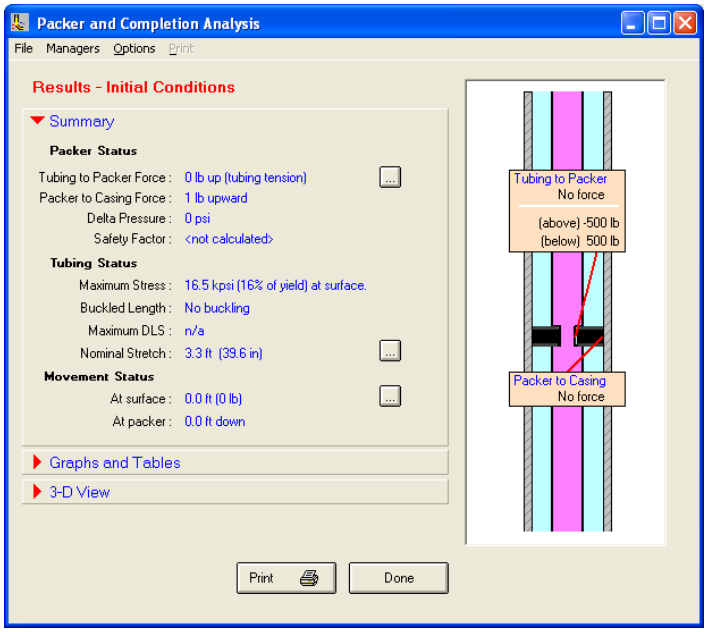
Click each related ellipses button to find/include information about the scenario and the various input data.

Conducting the Analysis

➢ To conduct an analysis

1. Click the **Analysis** button  to initiate calculations.

 The Results – Initial Conditions display screen appears.



2. Review/print the Summary (default) information, including Packer Status, Tubing Status, and Movement Status.

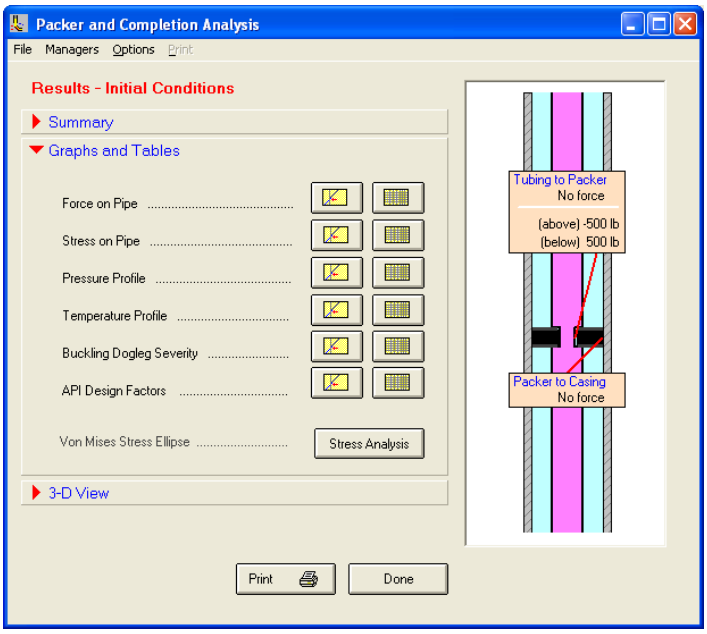
 ➢ To view details for each Summary category

 a. Click the category's related ellipses button

3. Review/print graphs, tables, and other related information generated from the calculations.

➢ To review/print the results graphs and tables

a. Select the Graphs and Tables node on the Results – Initial Conditions display screen.



b. To view each graph, click on its related Graph button .

c. To view each table, click on its related Table button .

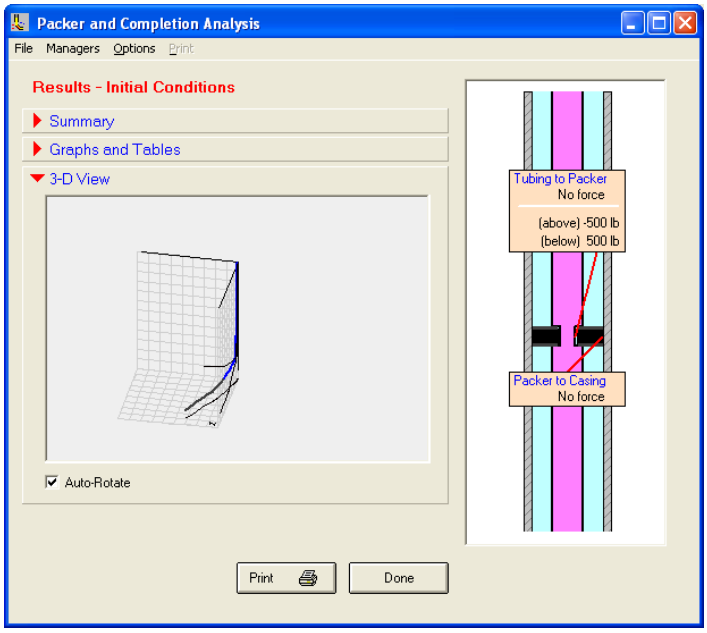
d. To view the stress analysis, click on the button by the same name.

e. From any graph, table, or analysis dialog, click the Print button to view and set up print options, and to print analysis results.

4. View the initial-condition results in 3-D.

➢ To view the initial-condition results in 3-D

a. Select the 3-D View node on the Results – Initial Conditions display screen.



Chapter 9
Using Solids Cleanout

This section describes the process involved in using Solids Cleanout, the Cerberus Hydra-based model for coiled tubing. Solids Cleanout calculates pump rates and the pressures involved in removing fill, such as sand, from a wellbore.

The Solids Cleanout Process

The process involved in using Solids Cleanout is as follows:

1. Create a new Solids Cleanout project or select an existing project (see page 201).



Solids Cleanout (along with the majority of other Cerberus models) is organized by projects.

2. Select well (see page 205).
3. Enter fill data (see page 206).
4. Select equipment (see page 206).
5. Select fluid data (see page 208).
6. Enter constraints (see page 209).
7. Calculate sensitivity (see page 210).
8. Enter pump rates for each stage (see page 211).
9. Adjust any other operating parameters (see page 212).
10. Perform the job-design calculation (see page 213).
11. Use/print the output data (see page 214).

Accessing Solids Cleanout (Hydra)

➢ To access Solids Cleanout from the Cerberus home page

- Select the Solids Cleanout button.

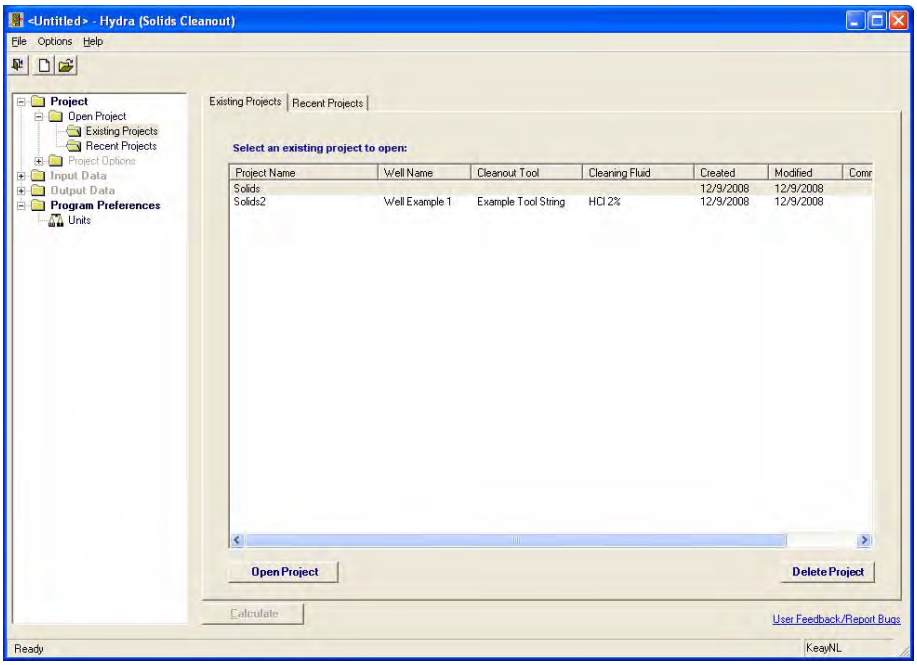


OR

- Select **Models > Solids Cleanout** from the menu bar.

The Solids Cleanout model loads, and the opening dialog of the Hydra (Solids Cleanout) model appears.

Solids Cleanout Opening Dialog



Working with Projects

The Solids Cleanout model is organized by projects.

Using the Solids Cleanout Opening-dialog Tools

The Solids Cleanout opening-dialog toolbar contains the following tools:

Solids Cleanout Main Toolbar

Tool Button	Tool Name	Function
	Exit	Exits Solids Cleanout.
	New	Opens the New Solids Cleanout Project dialog to create a new project.
	Open	Opens the Select Job dialog box to open an existing job.
	Save	Saves the current project to disk.

Setting Program Preferences

➢ To set program preferences

- Select **Options** > **Units** from the menu bar.

 OR

- Click + to expand the tree's **Program Preferences** node; click **Units**.

Creating a New Solids Cleanout Project

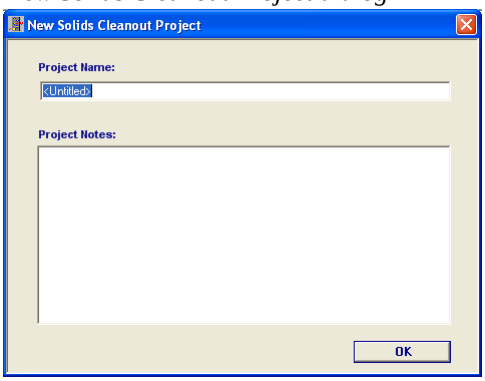
The following procedure outlines how to create a new Solids Cleanout project.

➢ To start a new project

1. From the Solids Cleanout Opening Dialog, choose one of the following:
 - Select **File** > **New** from the menu bar
 - Click the **New** toolbar button: 

 The **New Project** dialog appears:

 New Solids Cleanout Project dialog

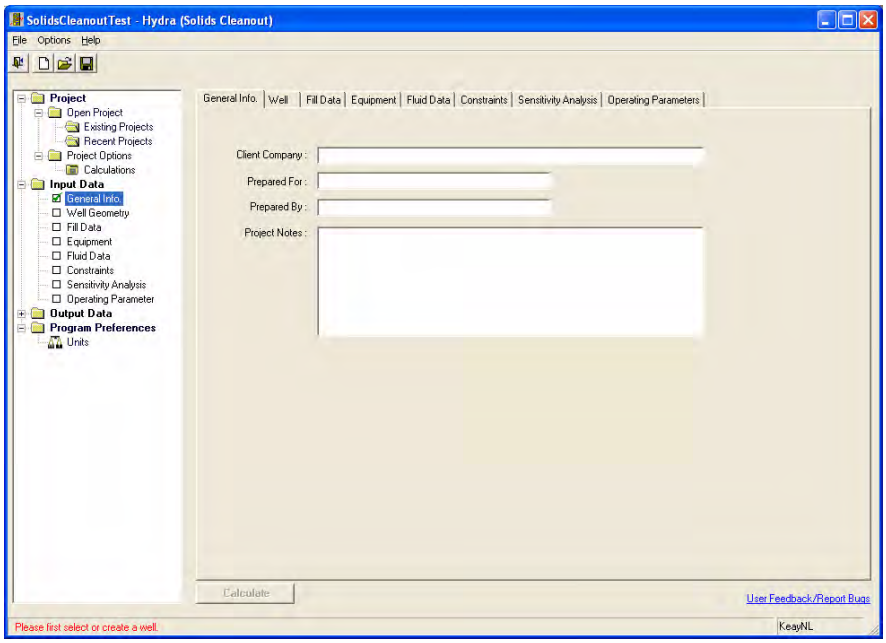




Start a new Solids Cleanout project directly from the Cerberus home page by selecting **File** > **New** > **Solids Cleanout Project**.

2. Provide a new name for the project. The name may consist of letters, numbers, and spaces, with a maximum of 32 characters.

The Solids Cleanout configuration display-screen appears.



3. Configure the project (see Entering Project-configuration Information for more information).

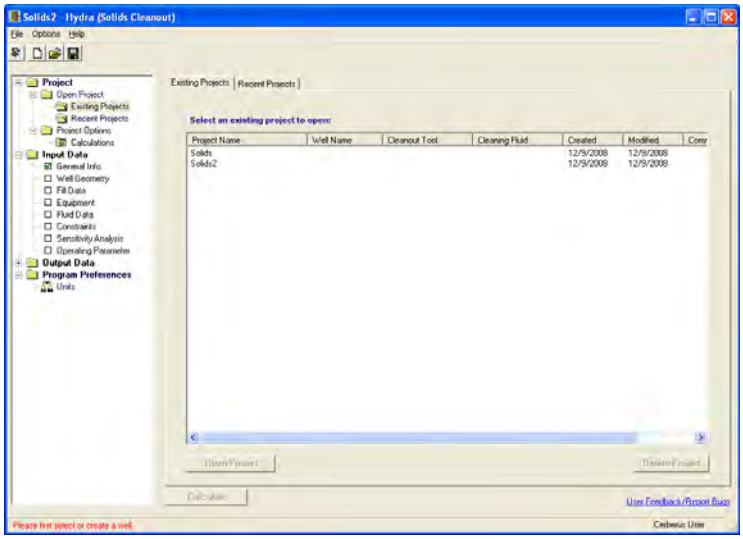
Opening an Existing Solids Cleanout Project

The following procedure describes how to open an existing project.

➢ To open an existing project

1. Choose one of the following from the Solids Cleanout opening dialog:
 - Select **File > Open** from the menu bar
 - Click the **Open Project** toolbar button: 
 - From the tree on the left-hand portion of the display screen, select **Existing Projects** or **Recent Projects** within the **Open Project** folder.

2. The following Hydra (Solids Cleanout) display screen appears, opened to the (default) Existing Projects tab.



3. Select the desired project within the tabpage on the right-hand portion of the display screen, and click the **Open Project** button in the tabpage.

From the Cerberus main menu, double-click the project name in the left-hand tree to open an existing project.

Saving a Solids Cleanout Project

The project configuration and last set of calculated values is retained when the project is saved. There are two ways a project can be saved:

- Click **Save** () on the toolbar.

 OR

- On the menu bar, click **File** > **Save** or **Save As...**.

Deleting a Solids Cleanout Project

When a project is deleted, the information is not recoverable.

➢ To Delete a Project

- From the menu bar, select **File** > **Delete Project**.
- From the Solids Cleanout Opening Dialog, click the **Delete Project** button.

Importing a Solids Cleanout Project

Import any project that is on removable media into Solids Cleanout.

➢ To Import a Project

1. If importing from removable media, insert the media in the drive.
2. Access the Solids Cleanout model.
3. From the menu bar on the Solids Cleanout Opening Dialog, click **File** > **Import...**.

Exporting a Solids Cleanout Project

Export any project in Solids Cleanout to removable media for importing to another computer with Solids Cleanout.

➢ To Export a Project

1. Access the Solids Cleanout model.
2. From the menu bar on the Solids Cleanout Opening Dialog, select **File** > **Export...**.

Entering Project-configuration Information

Configuration information in the Solids Cleanout model is accessible/editable via The Hydra (Solids Cleanout) display screen. View/edit the information by clicking the associated tab, or by selecting the related item in the tree.

Configuration information is stored within a project, so it is quickly accessible without the necessity of re-input.

Select a Well

➢ To select a well

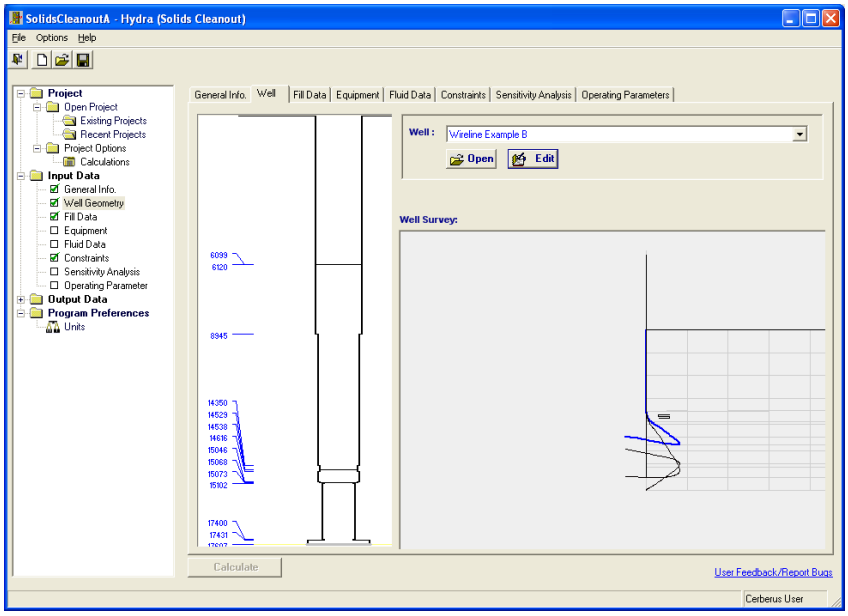
1. Within the Hydra (Solids Cleanout) display screen:
 - Click the **Well** tab.

 OR

 - Select **Well Geometry** from the tree.
2. Use the **Well** dropdown to select an existing well.

 OR

Click the associated **Open** button to access the Select Well dialog.



3. (Optional) To edit the selected well, click the associated **Edit** button. The Well Editor appears.

Enter Fill Data

➢ To enter fill data

1. Within the Hydra (Solids Cleanout) display screen:
 - Click the **Fill Data** tab.

 OR

 - Click (to check) **Fill Data** from the tree.
2. Enter the properties of the fill material and the calculation options.

Select Equipment (Reel, Coiled Tubing, and Tool String)

➢ To select equipment

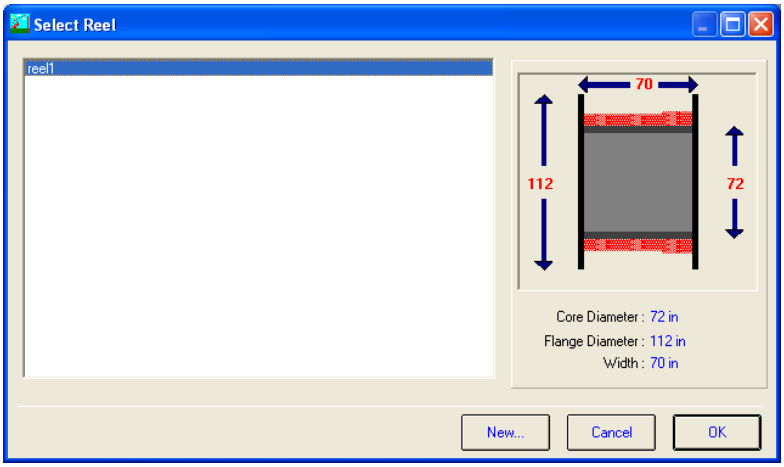
1. Within the Hydra (Solids Cleanout) display screen:
 - Click the **Equipment** tab.

 OR

 - Select **Equipment** from the tree.
2. Select the reel.

a. Click the Open button .

The Select Reel dialog appears:



b. Select the desired reel.

OR

Click the **New** button to create a new reel. The Reel Manager dialog appears.

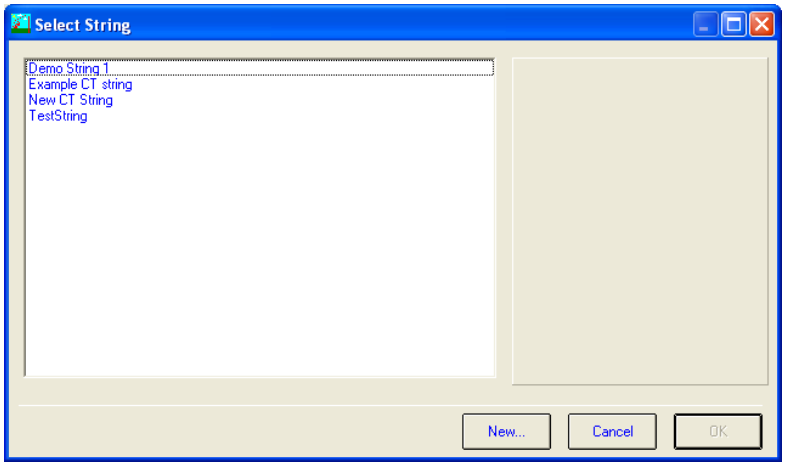
c. Edit the properties of the selected reel.

Click the **Edit** button associated with the reel to access the Reel Manager.

3. Select the coiled tubing.

a. Click the Open button .

The Select String dialog appears:



b. Select the desired string.

OR

Click the **New** button to create a new string. The String Editor appears.

c. Edit the properties of the selected string.

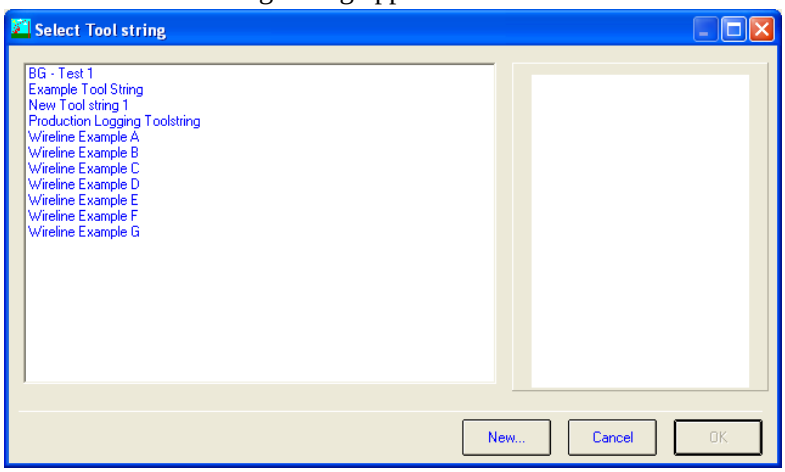
Click the **Edit** button associated with the string to access the String Editor.

4. Select the tool string (BHA).

 a. Click the Open button 

.

 The Select Tool String dialog appears:



 b. Select the desired tool string.

 OR

 Click the **New** button to create a new tool string. The Tool String Editor appears.

 c. Edit the properties of the selected tool string.

 Click the **Edit** button associated with the tool string (BHA) to access the Tool String Editor.

Select Fluid Data

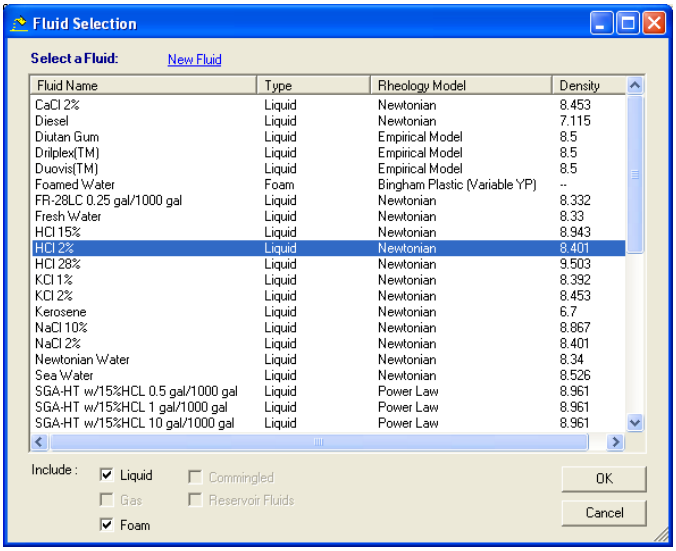
➢ To select fluid data

1. Within the Hydra (Solids Cleanout) display screen:
 - Click the **Fluid Data** tab.

 OR

 - Select **Fluid Data** from the tree.
2. Select the job fluid(s).

a. Click the **Open** button associated with each fluid type to access the Fluid Selection dialog.



b. Check/toggle the fluid type(s) to view (liquid, gas, commingled, or reservoir fluids).

c. Select the desired fluid.

3. Edit the properties of the selected fluid(s).

a. Click the **Edit** button associated with each fluid type to access the Fluid Manager dialog.

Enter Constraints

➢ To enter constraints

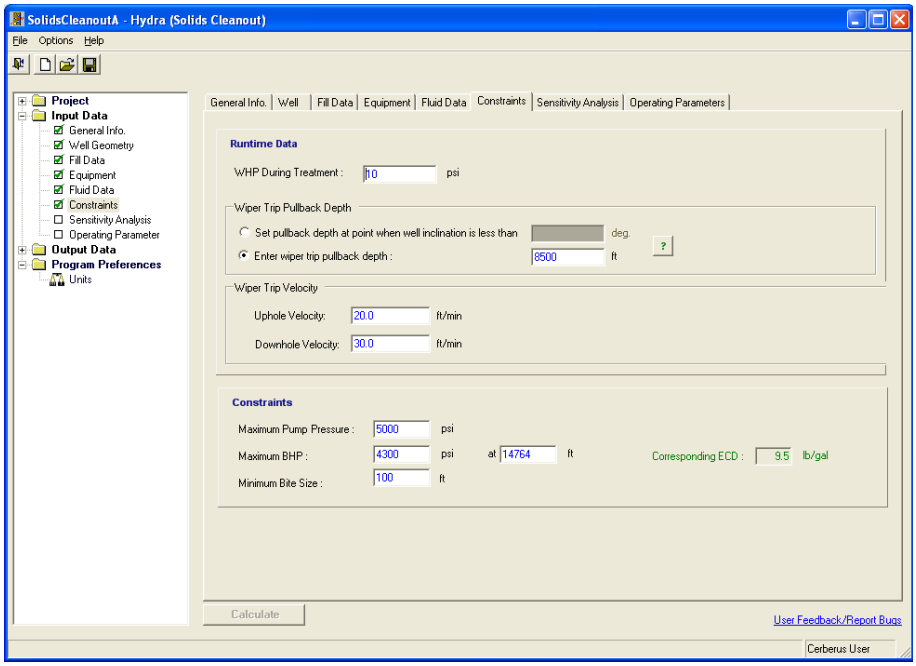
1. Within the Hydra (Solids Cleanout) display screen:

- Click the **Constraints** tab.

OR

- Select **Constraints** from the tree.

2. Enter any constraints that would help to make the calculated outcome more accurate.



For a deviated well, the Wiper Trip Pullback Depth can be determined either by entering the wiper trip pullback depth directly or by specifying the well inclination angel from which the wiper trip pullback depth can be calculated based on well survey data.

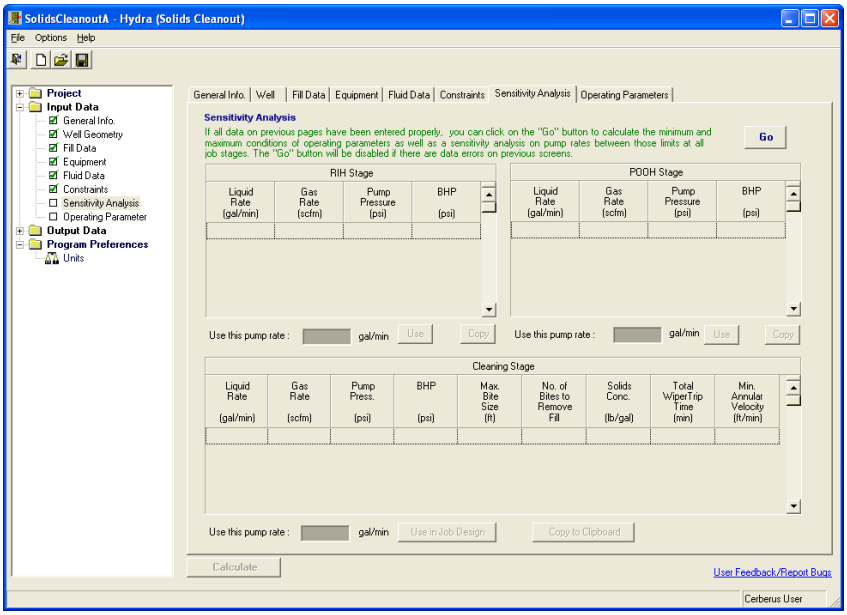
Calculate Sensitivity

➢ To calculate sensitivity

1. Within the Hydra (Solids Cleanout) display screen:
 - Click the **Sensitivity Analysis** tab.

 OR

- Select **Sensitivity Analysis** from the tree.



2. Click the **Go** button to calculate:

 a. minimum and maximum conditions of operating parameters.

 b. a sensitivity analysis on pump rates between the above limits at all job stages.

If the Go button is disabled, data errors are present within other Input Data screens.

Also, if the sensitivity calculation determines that a solids-cleanout operation would be unsuccessful given the provided parameters, a popup appears describing possible actions to take to correct the situation.



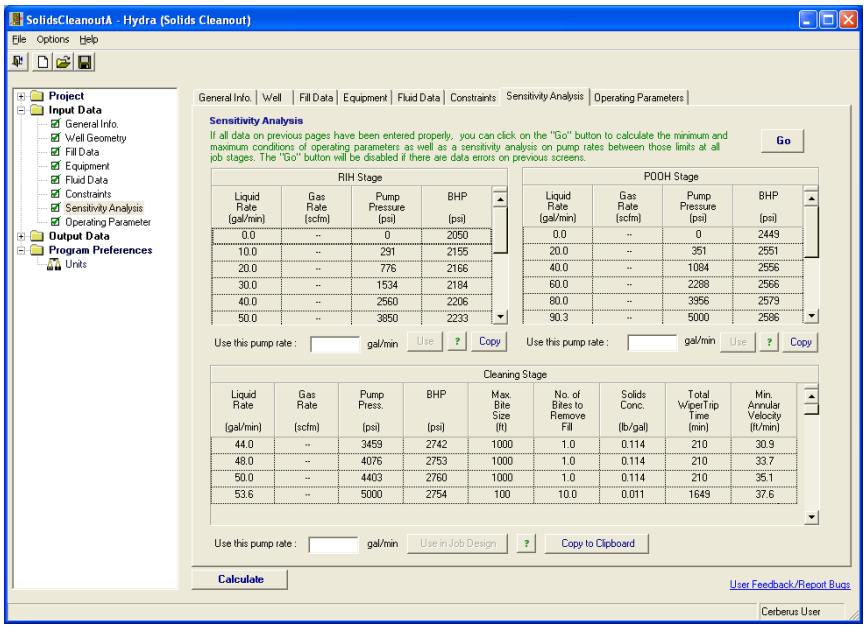
Enter Pump Rates for Each Stage

After analyzing sensitivity, enter pump rates for each stage (from the Sensitivity Analysis tab), in preparation for the final calculation step.

Although the Calculate button appears within each tab, it is prudent to wait until all parameters are properly entered before calculating.

➢ To enter pump rates for each stage

1. Within each stage (RIH, POOH, and Cleaning), observe the Liquid Rate (gal/min) in relation to the Pump Pressure, BHP, and other available parameters.



2. Determine a pump rate (Liquid Rate (gal/min)) that satisfies both Pump Pressure and BHP requirements for use in the final calculation.

 a. Choose the Pressure and BHP numbers in the table row that best correspond to the situation at hand, and observe the associated Liquid Rate.

3. Input the Liquid Rate that corresponds to the Pressure and BHP table-numbers within the **Use this pump rate** field underneath each stage's table.

 OR

 Input a value that is not included in the table. In this case, a new row is calculated within the table.

4. Click **Use** or **Use in Job Design**.

 Each value "used" is automatically entered in the Operating Parameters tabpage for use in the job-design calculation.

5. (Optional) Click **Copy** or **Copy to Clipboard**.

 The table-data is copied to the clipboard for use in an electronic spreadsheet.

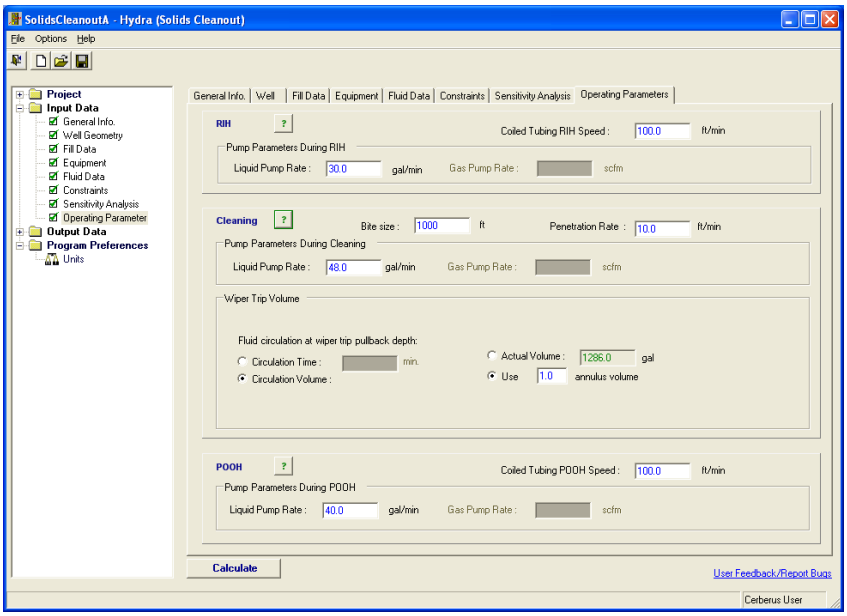
Adjust Any Other Operating Parameters

➢ To adjust any other operating parameters

1. Within the Hydra (Solids Cleanout) display screen:
 - Click the **Operating Parameters** tab.

OR

- Select **Operating Parameter** from the tree.



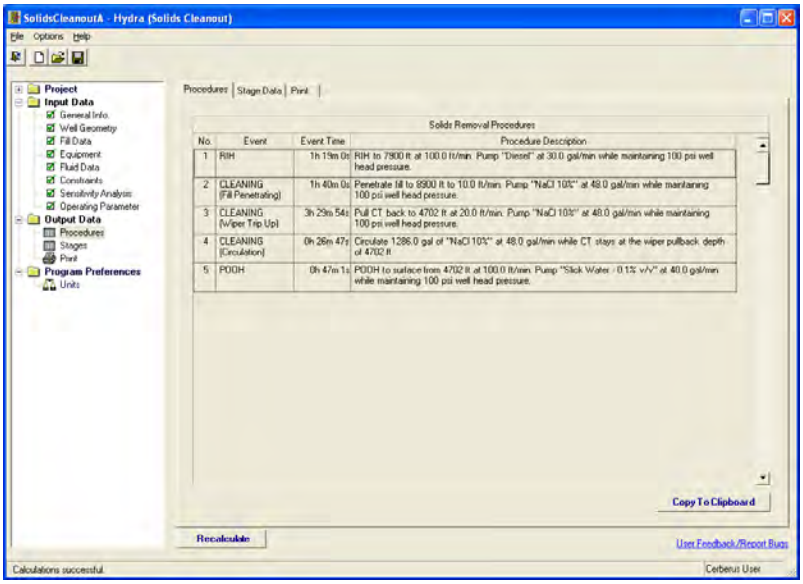
2. Adjust parameters as needed.

Perform the Job-design Calculation

➢ To perform the job-design calculation

1. After all parameters are in place as desired, click **Calculate** from any of the Solids Cleanout tabs.

2. If all parameters contribute to a successful calculation and solids-cleanout operation, the Output Data becomes available, as follows:



Using the Output Data

➢ To view the solids removal procedures (default)

Click the Procedures tab

OR

Select **Output Data** > **Procedures** from the left-hand tree.

➢ To view the solids removal stages

Click the Stage Data tab

OR

Select **Output Data** > **Stages** from the left-hand tree.

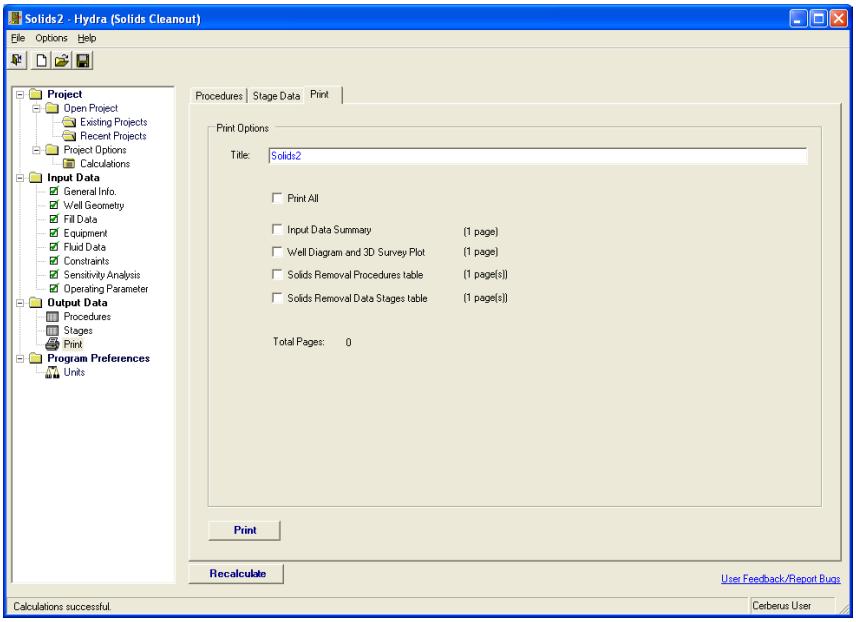
➢ To print the solids removal data

1. Click the Print tab

 OR

 Select **Output Data** > **Print** from the left-hand tree.

2. Toggle the options desired.



3. Click the **Print** button.

Chapter 10
Using the String Editor

String Editor is used to configure all the parameters that describe a coiled tubing string. String Editor stores each string in a different database. String inputs apply only to the tubing itself. The reel where the tubing is spooled is considered separately because a string can be moved from one reel to another during its working life.

- "Starting String Editor" on page 215.
- "The String Editor Display Screen" on page 216
- "Working with Strings" on page 220
- "Receiving a String on Removable Media from a Manufacturer" on page 237.
- "Using CT Specifications" on page 237
- "Configuring Strings" on page 238.
- "String Utilities" on page 251.
- "String Services" on page 254.
- "String History" on page 263.
- "String Fatigue" on page 264
- "Troubleshooting in String Editor" on page 267.

Starting String Editor

The following methods are available to start the String Editor:

- Orpheus and Hydra models include direct access to the String Editor via the String Editor toolbar button . In Reel-Trak (Job Manager), a String Editor icon is associated with the **String** field (in the Job Setup tab) when/if a string is selected.
- Within the Cerberus main menu, select **Managers > String Editor** from the menu bar.

String Editor opens in Simple Mode (showing a minimum of options) or Expanded Mode (showing a full range of options), depending on the following:

- If launched from Orpheus or Hydra, String Editor opens in Simple Mode.
- If launched from Job Manager, String Editor opens in Expanded Mode.
- If launched from the Cerberus main menu, String Editor opens in Expanded Mode if fatigue modeling capabilities are included in the user license.

➢ To manually select the String Editor mode

1. Select **Options** > **Expanded/Simple Mode** from the main String Editor menu bar.

Upon launching of String Editor, the editor opens with the selected string loaded. If String Editor is launched without a string selected, the Select a String dialog appears.

The String Editor Display Screen

The String Editor display screen is used to create new strings, open existing strings, and to manage strings after their creation.

The string name is shown alongside the display-screen name in the title bar.

String Editor Toolbar



Rest the cursor over a tool button for a few seconds to display the tool name.

Below is a description of the buttons on the main String Editor toolbar, along with each button's function.

String Editor Main Toolbar

Tool Button	Tool Name	Function
	Exit	Exits String Editor.
	New	Opens the New String dialog to create a new string (see page 220).
	Open	Opens the Select a String dialog to open an existing string (see page 228).
	Save	Saves the current string (see page 230).
	Delete	Deletes the current string from the hard drive (see page 231).
	Print	Opens the String Print dialog to print the current string (see page 234).

String Editor Main Toolbar

Tool Button	Tool Name	Function
	Sections	Opens the Sections properties screen to configure sections (see page 242).
	Welds	Opens the Welds screen to configure welds (see page 246).
	Zones	Opens the Zones screen to manage zones (see page 277).
	Lock	Allows the user to lock the string (see page 232).
	Fatigue Profile	Opens a fatigue graph for the string (see page 240).
	History	Opens a history log for the string (see page 234).
	Corrosion	Opens the Corrosion screen for the string, from which the user can choose between using the Wall Reduction or the String Life Reduction method (see page 250).
	Cut String	Opens the Cut String screen (see page 255).
	Splice String	Opens the Splice String screen (see page 258).
	Reverse String	Opens the Reverse String screen, with options of reversing the current string (no fatigue included) or reel-to-reel spooling operation (fatigue added to string) (see page 254).
	Weight	Opens the String Weight dialog showing an estimate of the string weight (see page 251).
	Volume	Opens the String Volume dialog showing an estimate of the string volume (see page 252).
	Cost	Opens the Summary screen with associated information for recording and estimating string cost (see page 220).

String Editor Main Toolbar

Tool Button	Tool Name	Function
	Help	Opens the Cerberus User Guide.

String Summary

The default view within a String Editor display screen is the string Summary information.

➢ To access the String Summary

1. Select the Summary button in the left-hand portion of the display screen (unless the Summary information already appears as the default).

 OR

2. Select **Edit > Summary** from the menu bar.

Properties

Select the **Properties** tab in the right-hand area of the display screen to access string-property information.

- **Tubing Type** - A dropdown from which the user may select from **Standard CT**, **Capillary**, or **FLATpak**™.



Tubing Type *is not visible by default. To view and use the* ***Tubing Type*** *field, select* ***Options > Show Tubing Type*** *from the String Editor menu bar.*

When the FLATpak tubing type is selected, fatigue is reset for the string. The Properties area expands to include additional parameters, from which the user may verify/alter the available properties.

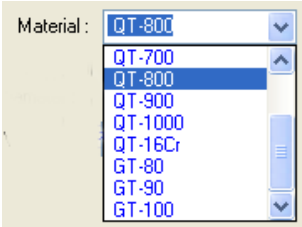


FLATpak is a jacketed/encapsulated CT product manufactured by CJS Coiled Tubing Supply and Source Rock Energy Partners.FLATpak can be used as a production conduit or to deploy downhole pumps, as well as other services. It consists of one or more individual CT and/or electric string encapsulated in a plastic/polymer jacket.

For this release of Cerberus, FLATpak strings are supported in Orpheus and Job Manager. FLATpak tubing that includes cable may not be defined.

- **Total Length** - Shows the length of the CT string.

- **Material** - A drop-down menu from which the user may choose from among various CT material types.



- **Diameter** - A drop-down menu from which the user may choose from among possible CT diameters. This information is automatically supplied by Cerberus when the string is selected.
- **Wall Size** - A drop-down menu from which the user may choose from possible CT string wall sizes. This information is automatically supplied by Cerberus when the string is selected.

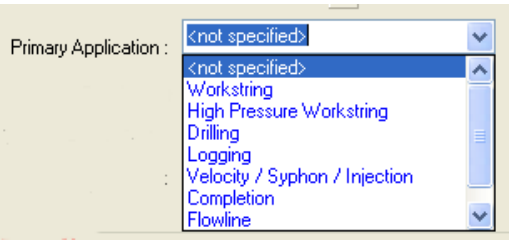
If a cable is installed in the coiled tubing, the user may click (to check) the **Cable installed** togglebox (associated with the cable icon  to include related parameters:

- **Cable type** - The type of cable installed in the CT.
- **Weight** - The cable weight in lbf/ft.
- **Cable diameter** - The cable diameter in inches.
- **Date installed** - The date on which the cable was installed in the CT.
- **H2S rated** - Click (to toggle) this field if the installed cable is H2S rated.

Info

Select the **Info** button to access secondary properties for a particular string. This action repopulates the screen to include the following:

- **Date Manufactured** - The date the CT was manufactured. To change this value, select the **Sections** button (in the left-hand portion of the display screen); then click the **Manufacturer Data** button and input the date in the **Date of Manufacture** field. The date then automatically populates the **Date Manufactured** field in secondary properties (**Info**).
- **Date Commissioned** - The date the CT string entered service. Change this value to the current date by clicking the associated ellipses button .
- **Primary Application** - The primary use for the CT string, such as Workstring, High Pressure Workstring, Drilling, and so forth. Choose from among the various options available in this drop-down menu.



- **Database** - The file name of the string.

- **Total Running Feet** - Automatically populated by fatigue calculations in Reel-Trak. In Cerberus, running feet is calculated only while running in-hole. For example, if the string is run in to 1000 ft, pulled out to 500 ft, then run back in to 1000 ft, and then pulled out of the well, the running feet will be 1500 ft.
- **Archived** - Indicates the archival status of the string. Click (to toggle) this field if the string is archived.

Actual Cost

Invoice Number and **Actual string cost (US$)** may be manually input in the two available fields.

String Diagram

A visual representation of the string is shown in the middle left-hand portion of the String Editor display screen.

The following may also be viewed:

Data Protection - Also present within the string diagram portion of the display screen, data-protection indicators show whether the string is protected (locked) or editable. Lock the string by clicking the associated lock button. When the string is locked, the button appears as a closed lock . When the string is not locked, the button appears as an open lock .

Working with Strings

Creating a New String



A new string can also be created by saving a copy of a string, either in its entirety or without the fatigue and history (see page 231).

Manually Creating a New String

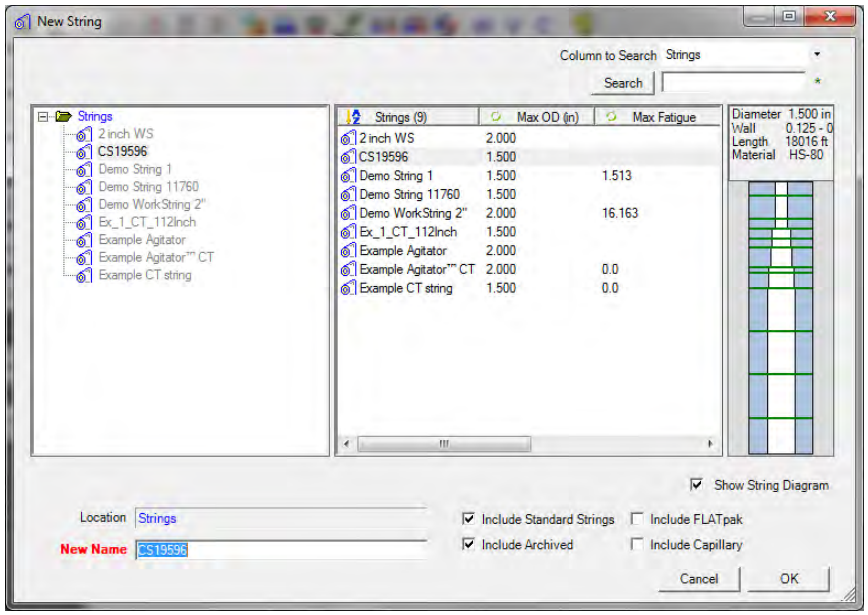
The following procedure outlines how to manually create a new string.

➢ To manually create a new string

1. Complete one of the following:
 - From the Cerberus main menu, select **Managers > String Editor** from the menu bar; then click the **New String** button on the Select a String dialog.
 - From the String Editor display screen, click **New**  on the toolbar
 - Select **File > New** from the String Editor menu bar.

 The New String dialog appears.

2. Click the “open folder” icon or the string-folder-name link beside the open-folder icon to select the folder in which to save the sting.



3. Enter the **New** String **Name**.

 The string name can be up to 32 characters, letters and numbers, including spaces.

4. Click (to check) **Include Standard Strings**, **Include Archived**, **Include FLATpak**, and/or **Include Capillary**, as appropriate.

5. (Optional) Click (to check) **Show String Diagram**.

6. Click **OK**.

7. **Configure** the string (see page 238).

Creating a String using the String Configuration Wizard

This wizard helps to create a new string when the configuration of the string is known. If help is needed to design a string for a particular application, use “Creating a String using the String Design Wizard” on page 222 instead.

➢ To create a string using the String Configuration Wizard

1. Select **Help > String Configuration Wizard** from the String Editor menu bar.

2. Enter a name for the new string. The name may be up to 32 characters long and may contain letters, numbers, and spaces. Then click **Next**.

3. Click (to toggle) the name of the supplier for this string. (Some parameters and specifications vary with each supplier. Clicking one of these toggles enters all the relevant supplier-specific information automatically.)

 When finished, click **Next.**



Cerberus supports Quality Tubing Inc.'s True Taper section-wall type.

4. Click (to toggle) the type of material to be used in the string. Click **Next** to continue.



When a material type is toggled, information about that material is displayed on the left-hand side of the wizard screen.

If Quality Tubing was selected in the previous wizard screen, the user must select QT-800, QT-900, or QT-1000 to use True Taper characteristics in wizard calculations.

5. From the String Diameter screen, select the diameter of the string from the dropdown. click **Next** to continue.



All available diameters for the chosen manufacturer are listed. It is assumed the diameter will remain constant along the entire length of the string.

6. The Straight or Tapered? screen appears. Click (to toggle) **Straight Wall** (indicating the wall is consistent along the length of the string) or **Tapered Wall** (indicating that the string has more than one wall size).

 a. If Quality Tubing's material type QT-800, QT-900, or QT-1000 was selected in Step 4, above, a togglebox becomes available, within which the user may: **Check this box if the string has any True Taper sections**.

7. Use the dropdown menu to select the **Number of Sections**. Click **Next** to continue.

8. Within the Section Properties dialog, complete the following:

 a. Input the **Section End** numeric value from the core end of the string.

 b. Select, from the dropdown menu, the **Weld Type** for the section described above.

 c. Select, from the dropdown menu, the **Nominal Wall Size** for the section.

 - If the togglebox **Check this box if the string has any True Taper sections** was checked on the Straight or Tapered? Quality Tubing-related screen, both an **Uphole Nominal Wall Size** and a **Downhole Nominal Wall Size** dropdown now appear for each section.

 d. For each additional section, input the information described in the step above and click **Next** to continue. The last section-end is used to determine the length of the string.

9. Click **Finish** to save the new string and exit the Wizard.

 The new string now opens.

Creating a String using the String Design Wizard

The String Design Wizard is used to create a new string to meet certain conditions or for a particular application. The wizard helps the user optimize the diameter, wall size(s), and material type for specific job conditions, such as in the design of a deviated well. The wizard also allows for using pre-designed data from Orpheus.

The String Design Wizard helps a user design a new CT string to meet expected conditions specific to a job. The user specifies the requirements, such as total road weight for the string and reel, flowrate, set-down weight, and pick-up force.

➢ To create a new string using the String Design Wizard

1. Select **Help > String Design Wizard** from the String Editor menu bar.

 String Design Wizard



2. Click (to toggle) a design method.

 The design methods include: **Vertical well** (recommended by NOV CTES); a **Simple deviated well**; or a **Specific job using pre-defined Orpheus well data**. Click **Next** to continue.

3. The Manufacturer dialog appears.

 a. Optionally toggle (to check) **Always use this string manufacturer when running String Design Wizard and String Configuration Wizard in future**.

 b. Select the manufacturer of the string. (Some parameters and specifications vary with each supplier.)

 The Job Parameters dialog appears. Alter parameters as needed.

4. Click **Next** to view the Additional Job Parameters dialog; enter additional job parameters as needed.

 The string should be designed to handle the most challenging well likely to be encountered in normal operations.

 Enter the **Desired string length**.

 Tubing remaining on the reel is disregarded in the Forces calculations, but included in the maximum weight and minimum flowrate calculations, when enabled.

 Click **Next** to continue.

5. The String Design Criteria screen appears.

 Click (to check) to enable associated fields and to specify any limitations pertaining to the design.

 These limitations include:

 - **Crane/road weight limitation** - A weight limit placed on the string, excluding the reel.
 - **Flow rate requirement** - Amount of water the string must be able to pump without exceeding an

entered pump pressure.

- **Must fit existing reel** - Lists the Core diameter, Flange diameter, and Width of an existing reel the CT must be able to fit onto. Click **Browse existing reels...** to select an existing reel for these values.
- **Include Collapse** - The ovality and safety factor percentage, plus the selection of the collapse model.

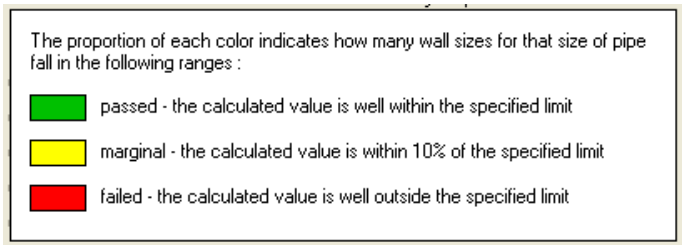
6. Click **Next**.

 The String Diameter screen appears.

7. Select the **diameter** of the string. Only one CT diameter is considered at a time.

 View the details by selecting **View Details...**.

 Select the Color Key button to see the following information about the colors appearing in the String Diameter dialog:



 Select **Remove failing sizes** to display only the wall sizes [whose calculated limit in relation to the toggled CT diameter] do not fail.

 Click **Next** to continue.

8. The Wall Sizes dialog appears.

 Select the wall size range.

 Select the smallest and largest acceptable tubing sizes for the job from the drop-down menus. In practice, the smallest and largest wall sizes are rarely used except for special applications.

 Click **Next** to continue.

9. The Materials screen appears.

 Click (to check) the desired material strength(s).

 The wizard attempts to design a string using each of the material strengths selected. If a material fails to yield a solution, the wizard will move up to the next strongest material and try again.

 Click any of the checkboxes to deselect that material from consideration by the wizard.

 Click **Next** to continue.

10. The Calculation Constraints dialog appears.

 Input the calculation constraints for this string.

 The calculation constraints include two categories: Standard and Advanced.

 Select (to check) the desired toggleboxes for each constraint.



The Advanced constraints are only included if selected under Step 2 of this procedure. These features require the complete version of Orpheus in Cerberus.

Click **Next** to continue.

The String Design Wizard screen appears.

Based on the input, the Wizard offers possible solutions that meet the job's criteria.

11. Click to select a possible string.

 The String Details portion of the String Design Wizard dialog appears, populated with all factors for the string as designed by the wizard.

 Click **Next** to continue.

12. The String Design Complete! portion of the String Design Wizard dialog appears.

 Strings can be reviewed and compared against the job criteria for the best match. To do this, click the **String Select Wizard...** button within the String Design Complete! portion of the String Design Wizard.

 After accessing the String Select Wizard, the closest matches are indicated by a green square, marginal results by a yellow square, and failing results by a red square.



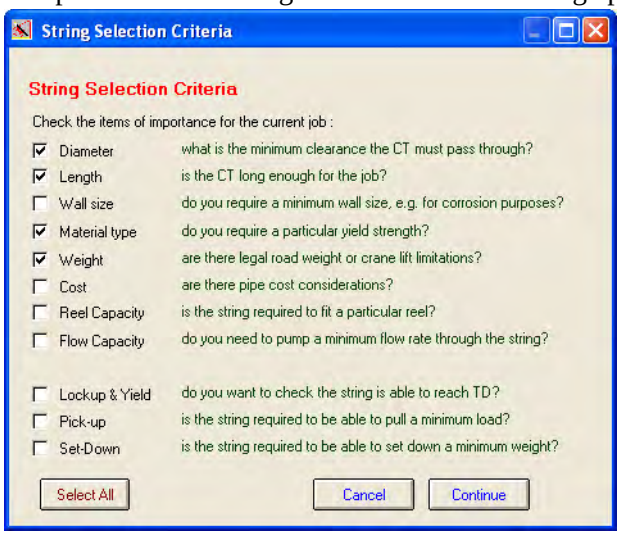
To customize the view of the available strings, select **View > Hide** *or* **View > Sort** *from the menu bar, and choose from among available options.*

13. To change the job input criteria, choose one of the following methods:
 - Click the **Criteria** button.
 - Select, from the menu bar, **Criteria > View All; Diameter; Length; Wall; or Material**.
 - Click to select a specific criterion from among the **Properties** on the right.

14. If a single criterion was selected, the String Selection Criteria dialog appears, specific to the selected criterion.

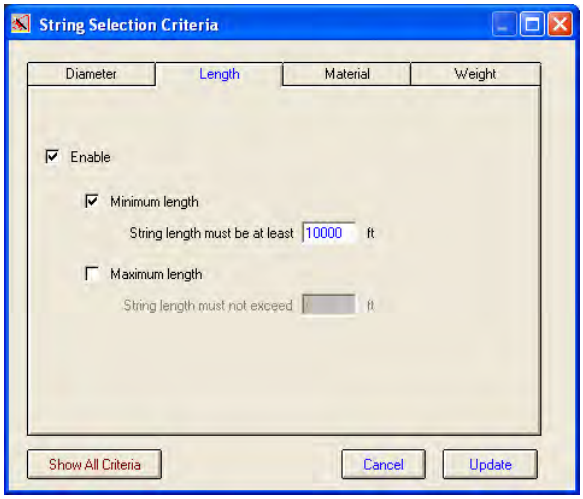
 Alter parameters as needed, and click **Update** to change that criterion within the design.

15. If the **Criteria** button or, from the menu bar, **Criteria > View All** was selected, the String Selection Criteria portion of the String Selection Criteria dialog appears, with all criteria available for selection.

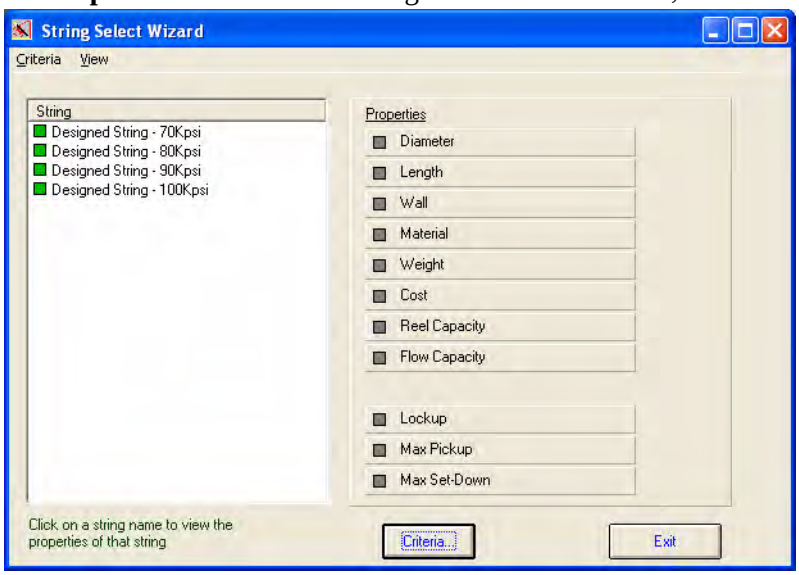


Click (to check) each item of importance; then click **Continue**.

a. Each selected criterion becomes available for adjustment within tabs, where the user may alter properties as needed.



- Click **Update** to return to the String Select Wizard . . . ,



- . . . or click **Show All Criteria** to return to the String Selection Criteria portion of the String Selection Criteria dialog. Here, the user may again click (to check) each item of importance and click **Continue** to weigh alternate criteria.

16. When criteria have been weighed successfully, click **Exit** to return to the String Design Complete! screen.

 A string MUST be selected and **Next** clicked to complete the String Design Wizard.

Note that a string must be saved before the details of the string can be printed or an order form for the string can be printed.

The user may:

- Save the string as a new database for future use;
- Print the string details;
- Print an order form for the string to fax or email the preferred supplier for a quote; or,
- Review and compare all strings by selecting the String Select Wizard.

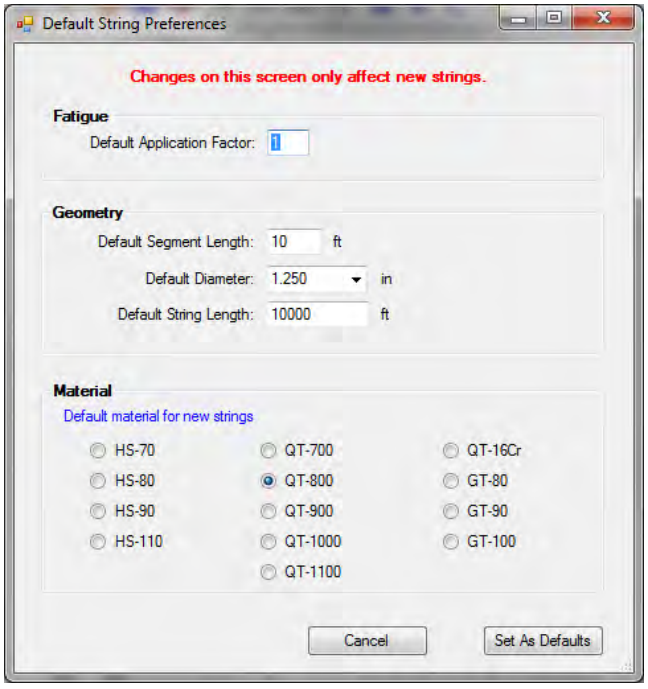
17. Click **Finish** to close the String Design wizard.

Default String Options

➢ To set the initial options for all newly created strings

1. Select **Options > Default String Preferences** from the menu bar.

 The Default String Preferences dialog appears.



2. Input the **Default Application Factor**. This is normally “1” unless it is company policy to modify fatigue by some amount.

3. Input/select the **Default Segment Length**, **Default Diameter**, and **Default String Length**.

4. Click (to toggle) the default **Material** for new strings.

5. Click the **Set As Defaults** button to finalize the settings.

Opening an Existing String

The user may open strings from the local database, import strings from removable media, or load strings from another computer on the network or from the Internet. Strings may be sorted according to various criteria.

If the desired string isn’t listed, it may be archived. Try changing the string editor preferences to show archived strings.

➢ To open an existing string

- Click **Open** on the String Editor toolbar.

 OR

- Select **File > Open** from the menu bar.

Importing a String

Sometimes, manufacturers supply a purchased string on removable media. If this is the case, the string must be imported into Cerberus. To copy a string to another computer, it must be exported onto removable media (flash drive, CD, and so forth) or network drive, then imported on the other computer.

➢ To import a string

1. If uploading from removable media, insert the removable media in the drive.
2. Start String Editor.
3. Select **File** > **Import** from the menu bar.

 The **Select a string project to import** dialog appears.
4. Select the drive and path where the string or strings are located.
5. Select the string or strings to upload.
6. Click **Open**.
7. Click **Yes** when the import is complete.

To . . .	Do this . . .
Select a group of items that are next to each other.	Click the first item, then hold down the SHIFT key and click the last item.
Select multiple non-adjacent items.	Hold down the CTRL key and click each item.

Exporting a String

The user may export strings from one computer to another, either via removable media, the network the computer is connected to, if any, or through the Internet, if an Internet connection is available.

When exported, the string(s) MUST be imported through the procedure outlined in “Importing a String” on page 229.



*The **Importing a String** procedure MUST be used before Cerberus recognizes a string database. The database **can NOT** be copied into the strings directory using Windows File Manager or Windows Explorer.*

➢ To export a string

1. If exporting to removable media, insert the removable media in the drive.
2. Start String Editor.
3. Select **File** > **Export** from the menu bar.

 The Select Strings dialog appears.
4. Select the string or strings to export.
5. Click (to check) export options: **Include Standard Strings** (default), **Include Archived** (default), **Include FLATpak**, and **Include Capillary**.
6. Click **OK**.

 The Save As dialog appears.
7. Select the destination drive and path.

 String backups are transferred when exporting a string to another computer. This allows correction of user errors on other computers.
8. Click **Save**.
9. Click **OK.**

Saving a String

Saving strings records any changes to the string in the database on the hard drive.

The changes are only applied to any new uses of that string. If the edited string is already in use, it will continue to use the old criteria.

➢ To save a string

Complete one of the following:

- Click **Save**  on the toolbar.
- Select **File** > **Save** from the menu bar to save a string with the same name.
- Select **File** > **Save As** from the menu bar to save the string under a new name.

Saving a String with or without Historical Data

➢ To save a string with or without historical data

1. Select **File** > **Save As** from the menu bar.
2. From the **Save Criteria** field in the lower portion of the dialog, use the dropdown to select S**ave With Historical Data** or **Save Without Historical Data**. See “String History” on page 263
3. Verify the method of saving the string selected in the previous step by visual examination.

 When the user selects **Save With Historical Data**, the current string backup is copied and renamed to the new string. See “To export a string” on page 230.

Saving a Copy of a String

Saving a string records changes to the string in the string database on the hard drive. Saving a copy of a string, except for the fatigue and history, is useful if the user wants to re-run the jobs a string has performed.

➢ To save a copy of a string

1. Select **File** > **Save As** from the menu bar.

 The Save As dialog appears.
2. Enter a new name for the string.
3. Select the save criteria.
4. Click **OK**.

 The string is saved under the new name.

Deleting a String

Use caution when deleting strings. When a string is deleted, the information cannot be recovered. Instead, consider archiving (see page 233) or retiring it.



Do NOT delete a string via Windows Explorer. A string should only be deleted using this procedure in Cerberus. Otherwise, Cerberus will indicate the string is still present and generate an error if a user tries to select and use that string.

➢ To delete a string

1. Start String Editor.
2. Click **Delete**  on the toolbar.

 The Delete Strings dialog appears.
3. Choose the string or strings to delete.

4. Click **OK**.

5. Click **Yes** to confirm deleting the string.

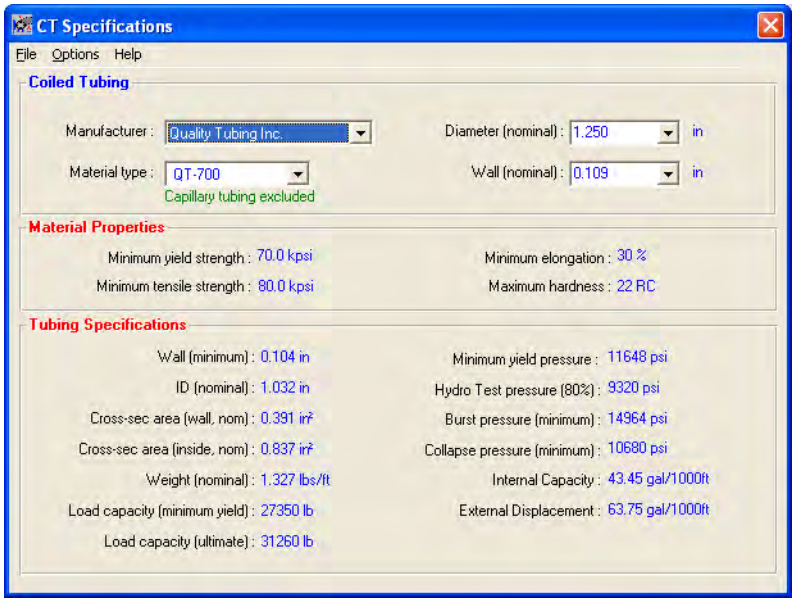
Using CT Specifications

Within CT Specifications, four parameters pertaining to the CT (manufacturer, material type, diameter, and wall (nominal)) may be altered by the user so that the software calculates the potential effects of these changes in material properties and tubing specifications.

➢ To use CT Specifications

1. Select, from the menu bar, **Help > CT Specs**.

 The CT Specifications dialog appears.



2. Using the dropdowns in the top portion of the dialog, select from the available parameters **Manufacturer**, **Material type**, **Diameter (nominal)**, and **Wall (nominal)**.

3. Observe the changes that occur in the **Material Properties** and **Tubing Specifications** as the above-described parameters are altered.

4. Determine the coiled tubing specifications most likely to correspond to the needed CT for a particular job.

Locking or Unlocking a String

Strings must be locked before a job can be run in execute mode in Reel-Trak. Locking a string also prevents inadvertent changes to the string database, although the string properties can still be viewed. Conversely, a string must be unlocked before any changes can be made to it.

A visual representation of the string is shown in the middle left-hand portion of the String Editor display screen.

➢ To lock or unlock a string

1. Click **Lock**  on the toolbar.

 OR

 Click the lock icon in the lower-left-hand portion of the visual-representation area (string-diagram area) of the string.

 OR

 Select **File > Locked** from the menu bar.

 A Warning dialog box appears stating a permanent entry will be recorded in the string history log. If the string is unlocked  it will become locked (unavailable for editing) when one of the lock methods is used. If the string is locked  , it will become unlocked (available for editing) when one of the lock methods is used.

Archiving a String

Strings that are no longer in use can be archived, or “retired.” When archived strings are hidden, they are no longer visible in the list of strings. However, the string database remains on the hard drive.

➢ To archive a string or remove a string from the archive **in String Editor only**

1. Select **File > Archived** from the **String Editor menu bar**.

 The check acts as a toggle; when **Archived** is checked, the string is archived. When **Archived** is not checked, the string is not archived.

➢ To generally archive a string or to remove a string from the archive

1. From the **Cerberus main menu**, select **Utilities > Base Manager** from the menu bar.
2. Select the Strings button; then click to select the string to archive.
3. Select **Utilities > Retire String** from the Base Manager menu bar.

The Retire String dialog appears.



4. Click (to toggle) one choice from each category. Removing a string and its associated jobs retains the string (and jobs) in the database.

 Strings and jobs that are removed are added back when the Project Tree and String Tree are rebuilt (from the main Cerberus menu bar, select **Utilities** > **Rebuild Tree**).

 Deleting a string removes the string and its associated jobs permanently.

5. Click **Retire String**.

Printing String Data

Print the status, configuration, used life, and history of a single string in String Editor.

Printing a Single String Report

The following procedure outlines how to print a single string report.

➢ To print string data to a printer

1. Click **Print** on the toolbar

 OR

 Select **File** > **Print...** from the menu bar.

 The String Print dialog appears.

2. Select/input Print Data.

 a. Leave the default Title as-is, input a new title, or delete the default title to have no title on the print-out.

 b. Click (to check) print options, as needed: **Status**, **Include Total Running Feet**, **Sections/Welds/Zones**, **History**, **Fatigue Graph** and **Diameter Graph**.

3. Select the Printer Options.

 a. Click the **Printer**-associated **Change** button to reveal a standard, Microsoft Windows®-like Print dialog.

 b. (Optional) Select the printer from the dropdown; select other options as needed and click **OK**.

 c. From the String Print dialog, input the number of **Copies** desired.

 d. To change the default NOV CTES logo, click the Change button.

 The Open dialog appears, from which the user may select a bitmap file to display as the printout logo.

4. Click the **Print** button.

In addition to printing to a printer, a PDF file can be printed for email distribution or for archiving.



To view a PDF file, the free Adobe Reader must be installed on the computer. It can be downloaded from Adobe's Web site at www.adobe.com.



NOV CTES does NOT recommend the use of any other PDF generator other than the CTES PDF Printer. Use of any other PDF generator can produce unpredictable and undesirable results.

➢ To use the NOV CTES PDF Generator

1. Click **Print**  on the toolbar

 OR

 Select **File** > **Print...** from the menu bar.

 The String Print dialog appears.

2. Select/input Print Data as described in Step 2, above.

3. Select the Printer Options as described in Step 3, above, EXCEPT, in Step 3b, select **CTES PDF Printer 25** from the Printer dropdown menu.

4. Click the **Print** button.

 A standard, Microsoft Windows®-like Save As dialog appears.

5. Input a file name.

 Navigate to the desired location where the file should be saved.

 Ensure that the **Save as type** selection is listed as "PDF file (*.pdf)". Otherwise, the file might not be saved correctly and might be unavailable for use.

6. Click the **Save** button.

Printing a Zoomed View of the String Life

➢ To print a zoomed view of the string life

1. Click **Fatigue**  on the String Editor toolbar.

 OR

 Select **View > Fatigue Profile** from the menu bar.

2. Select the scale ratio for the graph; (optional) select other plot options.

 a. Right-click the graph to adjust the scale of the string diagram for the printout.

 Zoom Out
 If the main Cerberus Setup dialog option **Auto-scale fatigue viewers** is clicked (checked), the option on the fatigue-profile Zoom Out menu is called **Zoom out (auto-scale)**.

 If the main Cerberus Setup dialog option **Auto-scale fatigue viewers** is not clicked (checked), the option on the fatigue-profile Zoom Out menu is called **Zoom out (100%)**.

 *Set the Auto-scale fatigue views option on the main Cerberus screen by selecting **Options** > **Cerberus Setup** from the menu bar.*

 Percentage Range
 Click to select from among a variety of percentage-range zoom options:
 0 - 100%, **0 - 50%**, **0 - 25%**, **0 - 10%**.

 b. For additional plot-option selections, right-click the graph and select **Options** to reveal the Plot Options dialog.

 - From the Fatigue screen, click (to check or to toggle) **Auto-scale fatigue viewer, Used life scale, Display high used life warning at** [input number] **%, Display fatigue due to corrosion**.
 - From the Diameter tab, click (to check) **Display high diameter warning,** under which the user may click (to toggle) **Percent growth: Nominal diameter +** [input number] **%**, or **Absolute growth: Nominal diameter +** [input number] **in**.
 - From the Options tab, click (to toggle) **Plot orientation** options **Plot from Core-end** or **Plot from Free-end**. Click (to toggle) **Plot style** options **Line** or **Filled area**. Finally, click (to check) **Display grid**, **Display string diagram**, and/or **Display zone diagram**.
 - Click **OK**.

3. Click **Print**  on the toolbar

 OR

 Select **File** > **Print...** from the menu bar.

 Complete printing steps as previously described.

Receiving a String on Removable Media from a Manufacturer

New string information received on removable media from the manufacturer should be imported.

➢ To receive a string from manufacturer's removable media

1. Import the string in String Editor (see page 229).
2. If the string arrived on a shipping spool and is being transferring to a work reel, reverse the string (see page 254).
3. If the string has a different segment length from the rest of the strings at the field location, change it to be the same (see page 239).

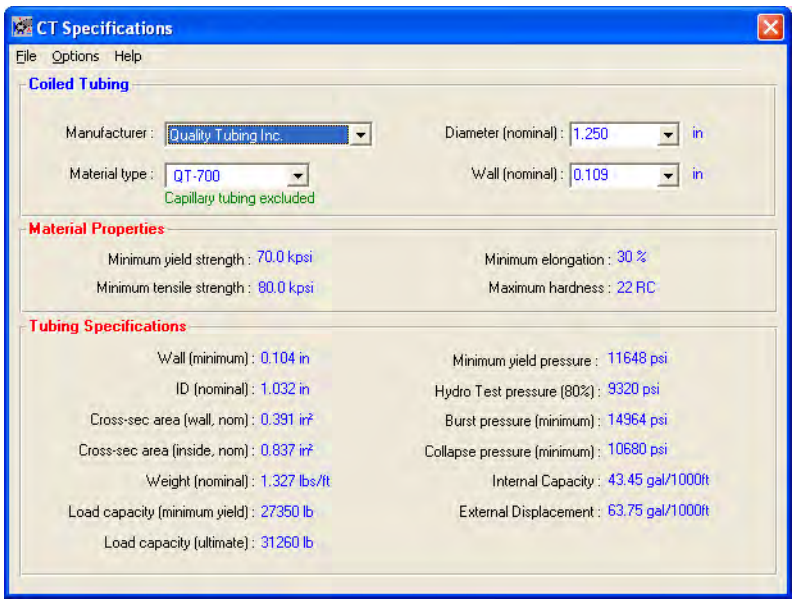
Using CT Specifications

Within CT Specifications, four parameters pertaining to the CT (manufacturer, material type, diameter, and wall (nominal)) may be altered by the user so that the software calculates the potential effects of these changes in material properties and tubing specifications.

➢ To use CT Specifications

1. Select, from the String Editor menu bar, **Help > CT Specs**.

 The CT Specifications dialog appears.



2. Using the dropdowns in the top portion of the dialog, select from the available parameters **Manufacturer**, **Material type**, **Diameter (nominal)**, and **Wall (nominal)**.
3. Observe the changes that occur in the **Material Properties** and **Tubing Specifications** as the above-described parameters are altered.

4. Determine the coiled tubing specifications most likely to correspond to the needed CT for a particular job.

Configuring Strings

Before configuring a string, obtain a basic understanding of the manner in which Cerberus deals with strings and the inputs required for various tasks. The following links direct the user to paragraphs that cover these topics.

- “Cerberus's View of a String” on page 238.
- “Required String Data” on page 240.

If configuring a used string, allow for existing fatigue (see page 240).

Create, configure, and delete strings using the Configuring Diameter procedure, which can be performed across the Cerberus software suite.

NOV CTES recommends configuring strings in the following sequence:

1. “Configuring Diameter” on page 240.
2. “Configuring Length” on page 241.
3. “Configuring Date Commissioned” on page 241.
4. “Configuring Sections” on page 242.
5. “Configuring Welds” on page 246.
6. “Configuring a Cable in String Editor” on page 247.

During the life of a string, 'flags' can be used on all or part of the string for special attention. For example, the user may develop concerns about particular areas with excessive corrosion or unrecorded bending cycles. Those areas can be identified as zones.

- “Configuring Zones” on page 248.

If tracking string corrosion is of interest, configure corrosion calculation options.

- “Configuring Corrosion” on page 250.

Cerberus's View of a String

When working with String Editor, the user must have a basic understanding of how String Editor looks at a string. The following links point to paragraphs that cover these topics.

- “What is Segment Length?” on page 239.
- “What is Position on the String?” on page 239.
- “What is Derating?” on page 239.

What is Segment Length?

Segment length is the smallest length of tubing for which used life (fatigue) is monitored. The smaller the segment length, the more accurate the fatigue profile. However, smaller segment length increases the number of calculations and the calculation time.

The default segment length of ten (10) feet (two (2) meters) should be adequate for the vast majority of applications. The smallest acceptable segment length is five (5) feet, or one (1) meter.

Understanding segment length is important to performing string operations. Strings can be spliced only as long as they have the same segment length.

A given job location must always use the same segment length and units to ensure strings can be spliced.

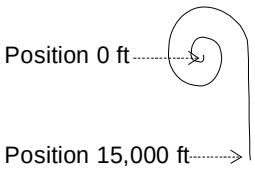
The total number of segments in a string is the string length divided by the segment length.

*To set units, select **Options** > **Units** from the main Cerberus menu bar. Then use the dropdown to select a unit set.*

What is Position on the String?

A position on the string is measured from the reel core end of the string. This method is used by the CT manufacturers. In a 15,000 foot string, the reel core end is at zero (0) feet, and the downhole end is at 15,000 feet. String position should not be confused with depth, which is a relative measurement measured from the free end of the CT and can vary, dependent on the zero depth datum.

In general, positions are measured from the reel core end. However, positions measured from both the reel core end and the downhole end are seen in the String Viewer.



What is Derating?

String life is calculated assuming plain pipe is in good condition. Welds and other physical features are known to have a lower fatigue life than plain pipe. Cerberus takes into account this reduced life with derating factors. Each weld is associated with a derating factor. Additional derating factors can be applied using zones.

Derating factors change the rate at which future fatigue accumulates.

A derating factor of 75% means that the fatigue life of that weld or zone is assumed to be at 75% of the base pipe. The fatigue life of the segment containing that weld or coinciding with that zone is derated *to* 75% (not *by* 75%).

Segments containing multiple derating factors are evaluated in the following manner:

- If the segment contains two or more welds (physically very unlikely), the worst-case derating factor is applied. (Weld derating factors are not additive.)
- If the segment coincides with two or more zones, both derating factors are applied in sequence. (Zone derating factors are additive.)
- If the segment contains a weld and coincides with a zone, the worst-case derating factor is applied.

A derating factor of 100% leaves the coiled tubing fatigue calculations unchanged. This should be used when flagging a zone for any reason.

Required String Data

Some inputs in String Editor are required for calculations, and some are optional. Which inputs are required and which are optional depend on what is in store for the string.

- For tracking fatigue, the diameter, length, sections, and welds information must be configured. The user may also want to configure zones.
- For exporting a string to Hydra for calculating tubing forces, the diameter, length, sections, and cable (if applicable) information must be configured.
- For calculating hydraulics in Hydra, the diameter, length, sections, and cable (if applicable) information must be configured.
- A cable for string weight and volume calculations can be configured in String Editor.
- Information regarding the string manufacturer and inspection records for welds can be entered. This information is not used in any calculations.

To quickly mock up a string to calculate tubing forces or hydraulics, most of the default values can be used. However, it’s recommend that all information for actual work strings be used.

Configuring Existing Fatigue on a Used String

When a used string is configured, existing fatigue on the string will need to be accounted for as well. The shorter used life can be accounted for by simulating jobs, by creating zones, or by using a combination of the two methods.

If the jobs that have been run on the string are known, a typical or similar job can be created in Job Manager, then run in Reel-Trak. The fatigue from the jobs is added to the string and future fatigue will accumulate at the normal rate.

If the job information for the string is unknown, zones can be added to the string. This method creates no fatigue, but increases the *rate* at which future fatigue accumulates.

Configuring Diameter

The diameter of a string is entered on the main String Editor screen. The diameter must be constant along the entire length of a string.

➢ To enter diameter

1. Open String Editor.
2. Open the desired string.

 The String Editor screen is repopulated to reflect the properties of the chosen string.
3. Select a string diameter.

 All available diameters from the selected manufacturer are listed in the dropdown menu.

Configuring Length

The length of a string is entered on the main String Editor screen.



Do NOT change the length of a real string that contains fatigue data through the main screen of String Editor. To cut a string, use the cut utility (see page 255). To splice a string, use the splice utility (see page 258).



When configuring a string, avoid changing the length (especially by shortening it) after sections, welds, and zones are configured. Changing the length can lead to loss of data or other unpredictable and undesirable results.

➢ To enter length

1. Open String Editor.
2. Open the desired string.

 The String Editor screen is repopulated to reflect the properties of the chosen string.
3. Enter the total string length in the **Total Length** field.

Configuring Date Commissioned

The date a string was commissioned is entered on the main String Editor display screen.

If a string is re-commissioned (the Date Commissioned field is changed), the new date is used in corrosion level calculations, if they are enabled. If an earlier date is entered, the next time a job is run in Reel-Trak, it calculates the additional corrosion for the days between the original commission date and the new commission date. If a later date is entered, the corrosion level is not reduced. Instead, Reel-Trak "waits" until the string has been in service for as long as it was in service before it was re-commissioned. For example, suppose a string was in service for 120 days when it’s re-commissioned. Reel-Trak waits until 120 days after the new commission date before adding more corrosion based on time in service. To immediately reduce the corrosion level, it can be manually reset.

➢ To enter the string commissioned date

1. Open String Editor.
2. Open the desired string.

 The screen is repopulated with the string's data.

3. Within the Summary screen, click the **Info**  button.
4. Input the date that the string was commissioned.

 Include all four digits of the year. To use the current date, click the ellipses  button.

Configuring Sections

Sections are configured within the Sections area of String Editor. Entering each length of tubing as a different section is unnecessary unless it will be used for tracking fatigue, at which point all sections must be configured for accurate fatigue calculations.

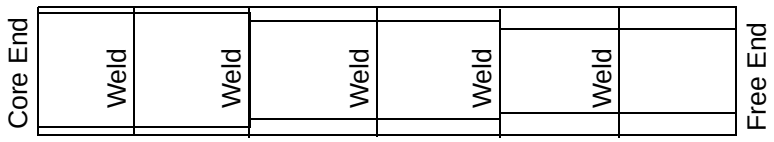
There are two ways to configure sections.

- "Configuring Sections by Wall Size" on page 243.
- "Configuring Sections by Strip" on page 244.

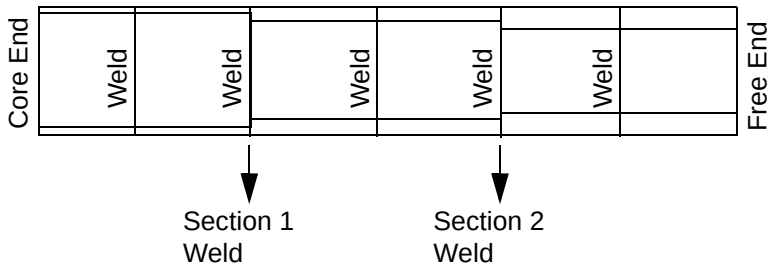
What are Sections?

A section is a length of the string with a single set of physical properties, such as wall thickness and material. Sections are configured within the Sections area of String Editor. There are two ways to configure sections. Each wall thickness may be configured as a section, or each strip between bias welds may be configured as a section. Although never recommended, it is technically possible to have sections with different coiled tubing materials, too. (For this reason, Cerberus uses the general name "sections" instead of "tapers".)

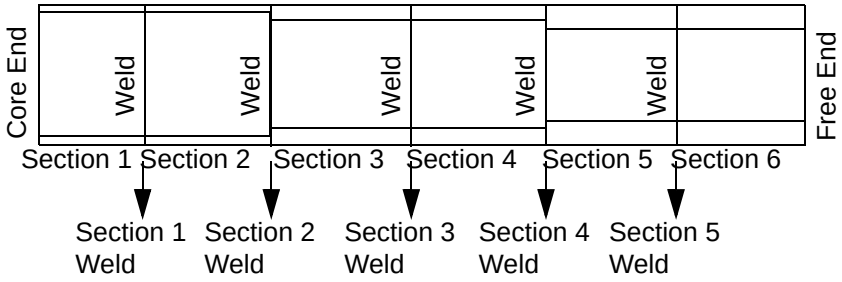
For example, consider the following string made out of six strips with three different wall thicknesses.



Configuring each wall thickness as a section results in three sections.



Configuring each strip as a section results in six sections.



Notice that each section, except the last one, is associated with the weld at its downhole end, which connects it to the next section. These welds are called section welds. By default, a section weld is a bias weld, but it can be changed to an orbital butt weld (machine) or a manual butt weld. If each strip is configured as a section, all the welds in a new string are section welds. If each wall thickness is configured as a section, the result will be some section welds and some other welds. These other welds would be entered on the Welds screen.

A section can have a single wall thickness, or it can have a continuous taper. For a section with a continuous taper, both the uphole and downhole effective wall thicknesses must be entered. The section will be tapered over the middle 80% of the section. The 10% at each end will be straight. (This matches how Quality Tubing, Inc. manufactures continuously tapered sections.)

Cerberus does not place any restrictions on what can be welded together; however, there are manufacturing limitations. Contact the manufacturer for string design limitations.

Entering only wall changes (tapers) as sections is the quickest and easiest way to configure a string. Entering all strips as sections takes longer, but provides for more detailed record keeping.

*To set a string as "tapered," open a string and select **Options** > **Tapered OD** from the menu bar.*

Configuring Sections by Wall Size

Sections may be configured by wall size.

➢ To configure sections by wall size

1. Complete one of the following:
 - Click the **Sections** icon  on the toolbar.
 - Select **Edit** > **Sections** from the menu bar.
 - Select the **Sections** button in the left-hand portion of the String Editor display screen.
2. Count the number of sections.

 On the manufacturer's specification sheet, pick out the bias welds which separate tubing with different wall sizes. These welds are section welds and separate different sections.
3. Set the number of sections.

 In the left-hand portion of the String Editor display screen, find and click the **Add Section** icon  as needed to increase the number of sections to the appropriate number.
4. Input the properties for **Section 1**.

 Section 1 starts at the reel core.

 Items that apply only to strips can be left blank, or values from any strip can be entered in the section.
5. To edit/input Manufacturer Data, click the **Manufacturer Data** button at the bottom of the **Section Properties** portion of the Sections screen.
6. Advance to the next section.

 To advance to the next section, use the arrows in the upper right-hand portion of the display screen or click within the appropriate section in the diagram located in the midsection of the Sections screen. Input the **Section Properties**.
7. Repeat the input of properties for each section. It is not necessary to enter the end position, length, or weld type for the last section.
8. To delete a section, click the **Delete** icon  on the toolbar or on the left-hand portion of the screen (Sections button).
9. To save the string, select **File** > **Save** (or **Save As**) from the menu bar.
10. Add the remaining welds on the manufacturer's specification sheet.

Configuring Sections by Strip

Sections may also be configured by strip.

➢ To enter each strip as a section

1. Complete one of the following:
 - Click the **Sections** icon  on the toolbar.
 - Select **Edit** > **Sections** from the menu bar.
 - Select the **Sections** button in the left-hand portion of the String Editor display screen.
2. Set the number of sections by increasing the number of sections to reflect the number of strips.

 In the left-hand portion of the String Editor display screen, find and click the **Add Section** icon  as needed to increase the number of sections (strips) to the appropriate number.
3. Input the properties for the first section (strip).

 The first section starts at the reel core.
4. To edit/input Manufacturer Data for this strip, click the **Manufacturer Data** button at the bottom of the **Section Properties** portion of the Sections screen.
5. Advance to the next section.

 To advance to the next section, use the arrows or click within the appropriate section in the diagram located in the midsection of the Sections screen. Input the **Section Properties**.
6. Repeat the input of properties for each section. It is not necessary to enter the end position, length, or weld type for the last section.
7. To delete a section (strip), click the **Delete** icon  on the toolbar or on the left-hand portion of the screen (Sections button).
8. To save the string, select **File** > **Save** (or **Save As**) from the menu bar.

Notes on Wall Reduction

The wall reduction factor is a string property used by Cerberus to compensate for the effects of acid and other damaging environments during a job. When a job is configured in Job Manager, the wall reduction is specified in the job type screen. The wall reduction is cumulative, so each job (that doesn't have zero wall reduction) contributes to the total wall reduction. The wall reduction is then subtracted from the minimum wall size to yield the effective wall size, which is used in calculations.

The wall reduction in String Editor can be viewed, but not changed.



Wall reduction is calculated using the same value along the entire length of the string. It is shown on the Sections screen so that the effective wall size can be seen. Wall reduction is calculated from the job type in Job Manager and the wall reduction option in Reel-Trak.

➢ To view wall reduction

1. Click **Sections** on the toolbar.

 The Sections screen appears, with Section Properties displayed on the right-hand side of the display screen.

 The wall reduction value appears within the **Wall size** properties.

Configuring Welds

Add welds, edit welds, and delete welds within the Welds area of String Editor. The string must be unlocked in order to add or remove welds. All non-section welds need to be entered in the string for fatigue calculations. Each weld may be a bias weld, orbital butt weld (machine), or manual butt weld.

Because welds are weaker than base pipe, each weld has a derating factor. String Editor provides recommended derating factors. The default weld derating factors are based on the conclusions of the 1995 Weld Joint Industry Project DEA-97. “Suggested Readings on Fatigue” on page 416, Item 16.) These default derating factors depend on the type of weld (manual butt, orbital butt, or bias), as well as whether or not the weld joins different wall thicknesses.

Adding a Weld

➢ To add a weld

1. Complete one of the following:

 - Click the **Welds** icon on the toolbar.
 - Select **Edit** > **Welds** from the menu bar.
 - Select the **Welds** button in the left-hand portion of the String Editor display screen.

 If the sections are configured, the section welds already appear in the list in the right-hand portion of the screen and on the string diagram in the mid-section of the screen.

 The welds are color-coded: red for Manual butt weld, blue for Orbital (CM) butt weld, and green for Bias.

2. From the **Welds** screen in the String Editor display screen, click the **Add Weld** icon.

 The Weld Properties area in the right-hand portion of the display screen becomes editable.

3. Enter the **Position of weld** from the reel core end.

4. Select, from the dropdown, the **Weld Type**.

 Weld types are bias weld, orbital butt weld, or manual butt weld.

5. Accept the default **Derating** factor, or input a new one.

 The new weld appears in the weld list and a color-coded line appears on the string diagram.

6. To save the string, select **File** > **Save** (or **Save As**) from the menu bar.

Editing a Weld

Welds in strings are editable.

To edit the position of a section weld, the sections must be reconfigured (see page 241).

➢ To edit a weld

1. Complete one of the following:
 - Click the **Welds** icon on the toolbar.
 - Select **Edit > Welds** from the menu bar.
 - Select the **Welds** button in the left-hand portion of the String Editor display screen.
2. From the list in the right-hand portion of the display screen available from the Welds screen, select the weld to edit.
3. Edit the **Weld Properties** as desired.

 The values for the **Weld Position**, the **Weld Type**, and the **Derating** factor can be changed.

 The **Weld Type** for a section weld cannot be changed.
4. To save the string, select **File > Save** (or **Save As**) from the menu bar.

Deleting a Weld

Welds can be deleted (removed) from the string, as well.

➢ To delete a weld

1. Complete one of the following:
 - Click the **Welds** icon on the toolbar.
 - Select **Edit > Welds** from the menu bar.
 - Select the **Welds** button in the left-hand portion of the String Editor display screen.
2. From the list in the right-hand portion of the display screen available from the Welds screen, select the weld to delete. Section welds cannot be deleted.
3. Click the **Delete** icon on the toolbar or on the left-hand portion of the screen (Welds button).

Configuring a Cable in String Editor

If a cable is installed in the string, it should be configured. The cable information is used in Hydra calculations of flow rate and frictional pressure drop. It is also used for string weight and string volume as well as calculations in Hydra.

For information about the NOV CTES cable injector, which can install cable with the string on the reel, please Contact Us

➢ To configure cable information

1. Select the **Summary** button in the left-hand portion of the String Editor display screen.
2. Click (to check) **Cable Installed** .
3. Input the cable information.

 This information includes the **Cable type**, **Weight** per length, **Cable diameter**, **Date installed**, and a checkbox to indicate the cable is **H2S rated**.

Configuring Zones

Zones are configured within the Zones screen of the String Editor display screen. Typically, zones are added to a used string; not a new string. Zones are created due to concerns for a part of a string that has suffered from excessive corrosion, damage that needs to be flagged, or because the used life of that portion needs to be changed.

What are Zones?

Zones are used to flag parts of the string that need to be watched, such as areas of corrosion or damage. Zones can also be derated if they have an uncertain fatigue history or if it is believed the life will be shorter for some reason.

Entering a derating factor other than 100% will change the used life of the zone for all subsequent calculations. Choosing a factor other than 100% is highly subjective and should be used with care.

The derating factor can be left at 100% to simply track the zone without affecting its used life. To change the used life, change the derating factor. A derating factor of 75% means that the fatigue life of that zone is assumed to be at 75% of the base pipe.

➢ To view existing zones in String Editor

1. Select the Fatigue icon in the toolbar
2. If zones exist and the plot option **Display effective zones in the string diagram** is selected, observe the shading in the **Derate Zones** area at screen-bottom.

OR

1. Complete one of the following:
 - Click **Zones** on the toolbar.
 - Select **Edit > Zones** from the menu bar.
 - Select the **Zones** button in the left-hand portion of the String Editor display screen.

2. If zones exist, observe their diagrams (in vertical orientation) just to the left of the string diagram. Select a zone to see its related information in the Zones (right-hand) area of the display screen.

Adding a Zone

The following procedure describes how to add a zone to a string.

➢ To add a zone

1. Select a string.

 If the string is locked, unlock it (see page 232).

2. Complete one of the following:

 - Click **Zones**  on the toolbar.
 - Select **Edit > Zones** from the menu bar.
 - Select the **Zones** button in the left-hand portion of the String Editor display screen.

3. Click the **Add** button in the top of the right-hand portion of the display screen.

 If no zones existed previously, various fields become available in which to input parameters. If a zone or zones existed previously, the **Current zone** number is incremented to accommodate the new zone.

4. Input the positions for the zone:

 The **Start position (Core end)** and **End position (free end)** of the zone are measured from the reel core end.

 The zone diagram (to the left of the string diagram) depicts the new zone.

5. Enter a derating factor.

 Leave the **Derate zone to** field at 100% to flag the zone without affecting fatigue life.

6. Select from the dropdown, or input, a reason for derating.

Removing a Zone

➢ To remove a zone

1. Access the Zones area as described in Adding a Zone, above.

2. Select the zone by clicking it in the zone diagram, or by clicking the up-or-down arrows associated with the **Current zone** field.

3. Click the **Remove** button in the top of the right-hand portion of the display screen.

Configuring Corrosion

Track corrosion levels for a string using a simple corrosion calculation. To track corrosion, the corrosion settings must be specified for each string on the Corrosion screen. The corrosion level is then increased accordingly when a job is run in Reel-Trak. The corrosion level can also be reset manually.



The suggested corrosion derating factors have not been validated experimentally.

About the Suggested Corrosion Derating Factors

Although the suggested corrosion derating factors are not based on experimental data, they have been based on some general assumptions. The suggested corrosion derating factors provide the following results:

- If a string remains in the yard without performing any jobs, after its first year the corrosion level would be 27%, based on time alone. After the second year, it would be 63%.
- If a string performed 50 (fifty) acid jobs (in wells without any H2S or CO2), its corrosion level would be 50%, not including how long the string has been in service.
- If a string performed 50 (fifty) jobs in wells that contained either H2S or CO2, its corrosion level would be 100%, not including the presence of acid or how long the string has been in service.

Entering Corrosion Settings for a String

Corrosion settings for strings are entered in the Corrosion portion of the string. The following procedure outlines how to enter these settings.



The corrosion calculations can be changed at any time in a string's life. Different strings can also have different corrosion calculations. However, NOV CTES recommends using the same settings for all strings to avoid confusion.

➢ To enter corrosion settings

1. Select a string.

 If the string is locked, unlock it (see page 232).

2. Select **Edit** > **Corrosion** from the menu bar.

 OR

 Click **Corrosion**  on the toolbar.

 Corrosion information appears in the right-hand portion of the String Editor.

3. Click (to check) the **Use Wall Reduction Method** and/or the **Use String Life Reduction Method** checkbox.

 If the **Use String Life Reduction Method** option is checked, various parameters become available.

4. Select the string life options checkboxes.

 The options affect how the string life is calculated.

- To manually reset the corrosion level, click the **Reset** button in the **Current Status** section. Enter the new corrosion level in the Corrosion Reset dialog and click **OK**.
- To adjust the string's **Time in Service**, click (to check) the associated checkbox. Then, input a percentage to **Decrease string life by for every 30 days elapsed**, **starting from** [click up/down arrows to select] **days after string is commissioned**. Finally, input the **Commission Date**.
- To decrease string life by a specified percentage according to job type:
 - Click (to check) **Decrease string life by** [input percentage] **% for** [click to toggle] **every acid job** or **every job**.
 - Click (to check) **Decrease string life by** [input percentage] **% if well has H2S**.
 - Click (to check) **Decrease string life by** [input percentage] **% if well has CO2**.

5. Click **Set as New Default** to make the current settings the default settings for new strings.
6. Click Restore Cerberus Defaults to return to the original default corrosion settings.

String Utilities

String Editor has several string utilities. The following links point to paragraphs that cover these topics in depth.

- "String Weight" on page 251.
- "String Volume" on page 283.
- "Estimating Remaining Trips" on page 252.

String Weight

Based on the geometry of the string, the string weight, excluding the reel, can be estimated. The estimate takes into account the wall thickness of tapered sections. The weight of the string while empty and while water-filled can be viewed. The weight of the cable, if there is a cable installed, can be taken in account as well.

➢ To estimate string weight

1. Complete one of the following:
 - Click **Weight**  on the toolbar.
 - Select **Utilities** > **Weight** from the menu bar.
 - In the left-hand portion of the String Editor, select the Utilities button; then click **Weight** .

 String Weight information appears in the right-hand portion of the display screen.

 The screen displays the total string weight when **Empty**, and when **Water-filled**, excluding the reel.
2. Select a value for the amount of water in the cable.

The user may use the up-and-down arrows associated with the **Water-filled** weight to adjust the **percent full** amount, which, accordingly, adjusts the **Water-filled** weight calculation.

3. (Optional) Click the **Include weight of cable** checkbox.

 The weight of the cable is included in the total weight calculations and the values in the screen are reconfigured accordingly.



This checkbox is available only if the Cable installed box is clicked and Weight information is entered in the Summary portion of the String Editor. See String Summary for more information.

String Volume

The volume the string can hold is estimated based on the nominal wall size and for the minimum wall size. The volume calculation can also take into account the cable volume (if one is installed) and wall reduction.

➢ To estimate string volume

1. Complete one of the following:

 - Click **Volume**  on the toolbar.
 - Select **Utilities** > **Volume** from the menu bar.

 String Volume information appears in the right-hand portion of the display screen.

2. Select the **Capacity** options.

 The capacity options include:

 - **Use nominal wall** - Bases the calculations on the nominal wall thickness.
 - **Use minimum wall** - Bases the calculations on the minimum wall thickness.
 - **Apply wall reduction** - Includes a reduction for previous jobs run using this string.
 - **Subtract cable volume** - Includes the volume required by a cable, if present.

 The Capacity and Displacement values are recalculated accordingly when the above options are selected/de-selected.

Estimating Remaining Trips

The number of trips in and out of the hole remaining on the string can be estimated based on the current maximum fatigue.



Use this estimate as a rough guide only. The estimate is based on a single set of conditions. Actual field conditions will produce a different amount of fatigue.

➢ To estimate the remaining trips on a string

1. Select **Utilities** > **Estimate Remaining Trips...** from the menu bar.

 The Estimate Remaining Trips dialog box appears.

2. Enter the **Reel Diameter**, **Arch Radius**, and **Pressure** for the trips.

3. Click (to toggle) either **Coiled Tubing Depth** (and input **Start** and **End** information) or **Working Depth**.

*Click the question-mark icons to learn more about **Coiled Tubing Depth** and **Working Depth**.*

4. (Optional) Click (to check) **Apply Application Factor**. See "Default String Options" on page 228 for additional information.

5. Click **Estimate Trips**.

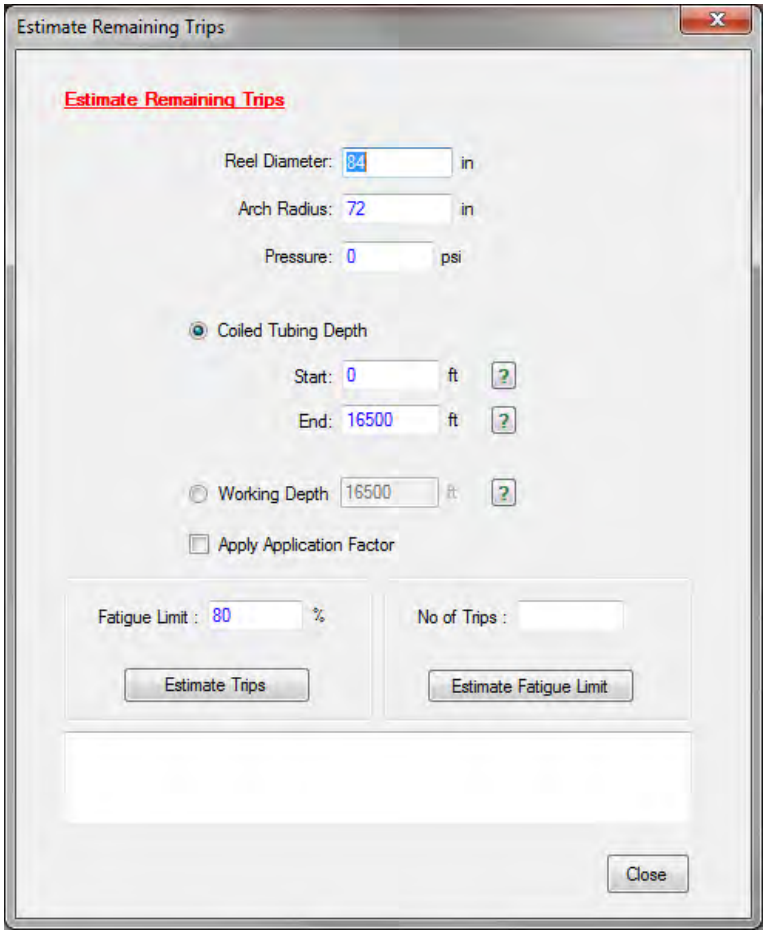
 The estimated number of remaining trips for the string is displayed, based on 80% fatigue life safety limit.

6. Input the Number **(No) of Trips** and click **Estimate Fatigue Limit** to estimate the fatigue limit.

7. Click **Close**.

 The Estimate Remaining Trips dialog is closed and the user is returned to the main String Editor display screen.

Estimate Remaining Trips



String Services

Cerberus organizes the reversing, cutting, or splicing of strings into "services."

When a string is reversed, cut, or spliced, that information is noted in String Editor. The following links point to paragraphs that cover each topic in more detail.

- "Reversing a String" on page 254.
- "Cutting a String" on page 255.
- "Splicing Strings" on page 258.

String services can be performed from the main Cerberus Project window when a project is selected, but not open in Reel-Trak.

This option is available if it is turned on. Turn on the String Services option by selecting ***Options > Cerberus Setup*** *and click (to check)* ***Display CT Services Control Panel*** *on the General tab.*

When a project is open in Reel-Trak, String Services are disabled in the main Cerberus Project window, but are available in the Job Setup screen of Reel-Trak if the job is not locked or executed.

Reversing a String

It is important to reverse the string in String Editor whenever the string is transferred to a different reel. For example, if the string arrives on a shipping spool and is transferred to a work reel, the string needs to be reversed. A string may also need to be reversed before splicing it to another string.

To include the associated fatigue to the string, Cerberus calculates fatigue for two bending events: unwrapping from the source reel and wrapping around the destination reel. Both bending events occur at zero pressure, and Cerberus calculates the bending radius for each event from the wraps of the string around the reel.

The tubing manufacturers configure a string according to the way it is shipped out of the factory. If the string arrives on a shipping spool and is transferred to a work reel, the string will need to be reversed.

Before transferring a string to a new spool, the reel's capacity can be estimated to see if the string will fit on it (see page 252).

➢ To reverse a string

1. Select a string.

 If the string is locked, unlock it (see page 232).

2. Complete one of the following:
 - From the Services screen, selected from the left-hand button on the String Editor display screen, select **Reverse String** .
 - From the menu bar, select **Edit > Reverse**.
 - From the toolbar, click **Reverse** .

 The Reverse String information appears in the right-hand portion of String Editor.

3. Select a spooling option:

- To reverse the string without recording the associated fatigue, click (to toggle) the Spooling Option **Reverse Only**.
- To reverse the string and calculate fatigue, click (to toggle) **Reel-to-Reel Spooling Operation**.

If **Reel-to-Reel Spooling Operation** is selected, enter the dimensions of the source reel and of the destination reel. The user can also click the reel icons to select reels configured in Reel Editor.

4. Click **Reverse**.

 String Editor reverses the string and notes the event (including whether fatigue was calculated and the user's login name) in the string history log.

Cutting a String

The string can be cut using the Cut String service, as follows:

- Cut off tubing from the free (downhole) end. This is done as part of routine maintenance (see page 255).
- Cut out tubing in the middle of the string. This is done to remove a damaged or highly fatigued portion of the string (see page 256).
- Cut string into two separate strings. This is done when cutting off part of a string and splicing it onto a different string, or to divide a long string shipped on a single spool from the manufacturer into two work strings (see page 256).

Cutting Off the End of a String

Tubing is cut from the free (downhole) end of a string as part of routine maintenance.

➢ To cut tubing from the free end of the string

1. Select a string.

 If the string is locked, unlock it (see page 232).

2. Complete one of the following:

 - Click **Cut**  on the toolbar.
 - Select **Edit > Cut** from the menu bar.
 - Select the Services button from the left-hand portion of the String Editor display screen; then click the **Cut String** icon.

3. Click (to toggle) **Cut and discard off free end of string (downhole)**.

4. Input the amount to cut.

5. Click the **Cut** button.

 A warning dialog box appears to confirm cutting the string.

6. Click **Yes** to confirm cutting.

 String Editor cuts the string and notes the event and the user's login name in the history log for the string.

Cutting a String in the Middle

To remove a damaged or highly fatigued portion of the string, the string is cut somewhere in the middle and the ends rejoined.

➢ To cut a string in the middle

1. Select a string.

 If the string is locked, unlock it (see page 232).

2. Complete one of the following:

 - Click **Cut**  on the toolbar.
 - Select **Edit** > **Cut** from the menu bar.
 - Select the Services button from the left-hand portion of the String Editor display screen; then click the **Cut String** icon.

3. Click (to toggle) **Cut and discard from middle of string**.

4. Input the beginning position of the cut **From** the reel-core end, in a multiple of the segment length.

5. Input the end position of the cut from the reel-core end (**To)**, in a multiple of the segment length.

6. Click (to toggle) the **type of weld** used to rejoin the string.

 The options are **Manual** or **Orbital**.

7. Enter a **Derating** factor for the weld, or accept the default.

8. Click the **Cut** button.

 A warning box appears to confirm cutting the string.

9. Click **Yes** to confirm the cut.

 String Editor cuts the string and notes the event and the user's login name in the history log for the string.

Cutting a String and Saving Both Parts as Separate Strings

To divide a long string shipped on a single spool from the manufacturer into two work strings, the string is cut and both parts saved.



Before transferring a string to a new spool, the reel's capacity can be estimated to see if the string will fit on it (see page 252).

➢ To cut a string and save both parts as separate strings

1. Select a string.

 If the string is locked, unlock it (see page 232).

2. Complete one of the following:

 - Click **Cut**  on the toolbar.
 - Select **Edit** > **Cut** from the menu bar.
 - Select the Services button from the left-hand portion of the String Editor display screen; then click the **Cut String** icon.

3. Click (to toggle) **Cut string and save as two separate strings**.

4. Input the position of the cut (**Cut at**).

5. Click the **String 1**-associated **Save As...** button to save the new first string.

 The String Description dialog appears.

6. (Optional) Enter a **description** for the new first string (or use the default existing-string name for the first string).

 (Optional) Click the **Show Existing...** button to view existing strings.

 Names can be a maximum of 32 characters, including letters, numbers, and spaces.

7. Click **OK**.

8. Click the **String 2**-associated **Save As...** button to save the new second string.

 The String Description dialog appears.

9. Enter a **description** for the new second string.

 Names can be a maximum of 32 characters, including letters, numbers, and spaces.

10. Click the **Cut** button.

 A warning box appears to confirm the cutting of the string.

11. Click **Yes** to confirm cutting.

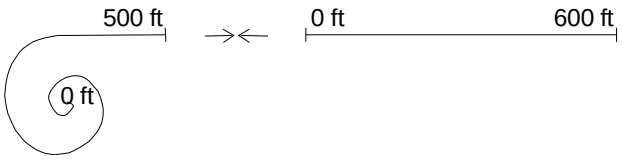
 String Editor cuts the string and notes the event and the login name in the history log.

 If the new string is put on a different reel, reverse it. (The downhole end becomes the reel core end (see page 254)).

Splicing Strings

Within String Editor, the splicing service joins two strings to create one new string. After the two strings are spliced together, three strings exist in String Editor: the two original strings and the third, spliced string. The two original string files can be archived.

When String Editor splices strings, it joins the free end of the main string to the core end of the second string. The second string may need to be reversed, depending on the ends being joined. The second string will not need to be reversed if it is transferred to a new reel, spliced, then spooled onto the main reel. There is no need to reverse the second string if it is spliced to the main string and spooled onto the main reel.



Strings can be spliced in several ways. The following links point to paragraphs that cover these ways in more detail.

- "Adding New Pipe to the End of a String" on page 258.
- "Adding New Pipe to the Middle of a String" on page 260.
- "Splicing Strings End to End" on page 261.
- "Splicing a String into the Middle of Another String" on page 262.

Only strings that are compatible with each other can be spliced.

Before transferring a string to a new spool, the reel's capacity can be estimated to see if the string will fit on it (see page 252).

Compatibility Requirements for Splicing Strings

Only compatible strings can be spliced together. In order to be compatible, both strings must be active (not archived) and must have the same values for the following items:

- units system,
- fatigue model,
- diameter, and
- segment length.

Adding New Pipe to the End of a String

When new pipe is added to the end of a string, the string stays on its reel and the new pipe gets spooled over it. One field weld is created where the new pipe is joined.

The new pipe will not have any fatigue damage. However, used pipe with fatigue can be spliced onto an existing string.

To ensure the spliced string will fit on the reel, the reel's capacity can be estimated (see page 252).

➢ To add new pipe to the end of a string

1. Select a string.

 If the string is locked, unlock it (see page 232).

2. Complete one of the following:

- Click **Splice**  on the toolbar.
- Select **Edit** > **Splice** from the menu bar.
- Select the Services button from the left-hand portion of the String Editor display screen; then click the **Splice String** icon.

3. Click (to toggle) **Add New Pipe** on the right-hand portion of the String Editor display screen.

 The screen is repopulated to show the New Pipe Properties section.

4. Input the **Splice Position** in the current string.

5. Input the properties of the new pipe.

 The properties include:

 - Length of New Pipe (must be a multiple of the segment length),
 - CT Diameter,
 - Material, and
 - Nominal wall thickness.

6. Within the Uphole Weld section of the display screen, select, from the dropdown, the **Weld** type.

 Weld types include **Manual** and **Orbital**.

7. Input the Derating Factor.

8. Click the **Splice** button.

 A warning dialog box appears to confirm the splice.

9. Click **Yes** to confirm splicing.

 String Editor splices the new pipe onto the string and notes the event and the user's login name in the history log for the string.

Adding New Pipe to the Middle of a String

When pipe is added to the middle of a string, the string is cut in two and the new pipe is welded in between. Two field welds are created, one on each end of the new pipe.



The new pipe will not have any fatigue damage. Splice in an existing string when adding used pipe with fatigue (see page 262).



To ensure the spliced string will fit on the reel, the reel's capacity can be estimated (see page 252).

➢ To add new pipe to the middle of a string

1. Select a string.

 If the string is locked, unlock it (see page 232).

2. Complete one of the following:

 - Click **Splice**  on the toolbar.
 - Select **Edit** > **Splice** from the menu bar.
 - Select the Services button from the left-hand portion of the String Editor display screen; then click the **Splice String** icon.

3. Click (to toggle) **Add New Pipe** on the right-hand portion of the String Editor display screen.

 The screen is repopulated to show the New Pipe Properties section.

4. Input the **Splice Position** in the current string.

 The splice position must be less than the length of the string.

5. Input the properties of the new pipe.

 The properties include:

 - Length of New Pipe (must be a multiple of the segment length),
 - CT Diameter,
 - Material, and
 - Nominal wall thickness.

6. Within the Uphole Weld section of the display screen, select, from the dropdown, the **Weld** type.

 Weld types include **Manual** and **Orbital**.

7. Within the Downhole Weld section of the display screen, select, from the dropdown, the **Weld** type.

 Weld types include **Manual** and **Orbital.**

8. Input the **Derating Factor** for each weld type.

9. Click the **Splice** button.

 A warning dialog box appears to confirm the splice.

10. Click **Yes** to confirm splicing.

 String Editor splices the new pipe onto the string and notes the event and the user's login name in the history log for the string.

Splicing Strings End to End

When strings are spliced end to end, the main string stays on its reel, and the second string is spooled over it. One field weld is created where the two strings are joined.



To ensure the spliced string will fit on the reel, the reel's capacity can be estimated (see page 252).

➢ To splice a string to the end of another string

1. If the core end of the SECONDARY string will become the free end of the spliced string, reverse the SECONDARY string (see page 254).
2. Open the MAIN string:
 a. Select a string.
 If the string is locked, unlock it (see page 232).
3. Complete one of the following:
 - Click **Splice** on the toolbar.
 - Select **Edit** > **Splice** from the menu bar.
 - Select the Services button from the left-hand portion of the String Editor display screen; then click the **Splice String** icon.
4. Click (to toggle) **Add Existing String**.
 The screen is repopulated with the Existing String Properties section.
5. Click **Select...** to select the second string.
 The Select String dialog appears, listing all available strings.
6. Select the desired **string**.
7. Click **OK**.
8. Within the Uphole Weld section of the display screen, select, from the dropdown, the **Weld** type.
 Weld types include **Manual** and **Orbital.**
9. Input the **Derating Factor** for each weld type.
10. Click the **Splice** button.
 A warning dialog box appears to confirm the splice.
11. Click **Yes** to overwrite the existing MAIN string, or click **No** to create a new string via the New dialog.
12. If No was clicked in the step above, input the **New string name**.
 String Editor splices the strings together and notes the event and the user's login name in the history log of the new string.

Splicing a String into the Middle of Another String

When a string is spliced into the middle of another string, the main string is cut in two and the second string is welded between the ends. Two field welds are created, one at each end of the second string.

To ensure the spliced string will fit on the reel, the reel's capacity can be estimated (see page 252).

➢ To Splice a String in the Middle

1. If the SECONDARY string is reversed during the splicing process, reverse the SECONDARY string (see page 254).
2. Open the MAIN string.
 a. Select a string.
 If the string is locked, unlock it (see page 232).
3. Complete one of the following:
 - Click **Splice**  on the toolbar.
 - Select **Edit** > **Splice** from the menu bar.
 - Select the Services button from the left-hand portion of the String Editor display screen; then click the **Splice String** icon.
4. Input the **Splice Position** of the first-string insertion point for the second string.
5. Click **Select...** to select the second string.
 The Select String dialog appears, listing all available strings.
6. Select the desired **string**.
7. Click **OK**.
8. Within the Uphole Weld section of the display screen, select, from the dropdown, the **Weld** type.
 Weld types include **Manual** and **Orbital**.
9. Within the Downhole Weld section of the display screen, select, from the dropdown, the **Weld** type.
 Weld types include **Manual** and **Orbital.**
10. Input the **Derating Factor** for each weld type.
11. Click the **Splice** button.
 A warning dialog box appears to confirm splicing the two strings.
12. Click **Yes** to confirm splicing.
13. To overwrite the existing string (default), click **Yes**. To create a new string name, click **No**.
 String Editor splices the strings together and notes the event and the user's login name in the history log of the string.

String History

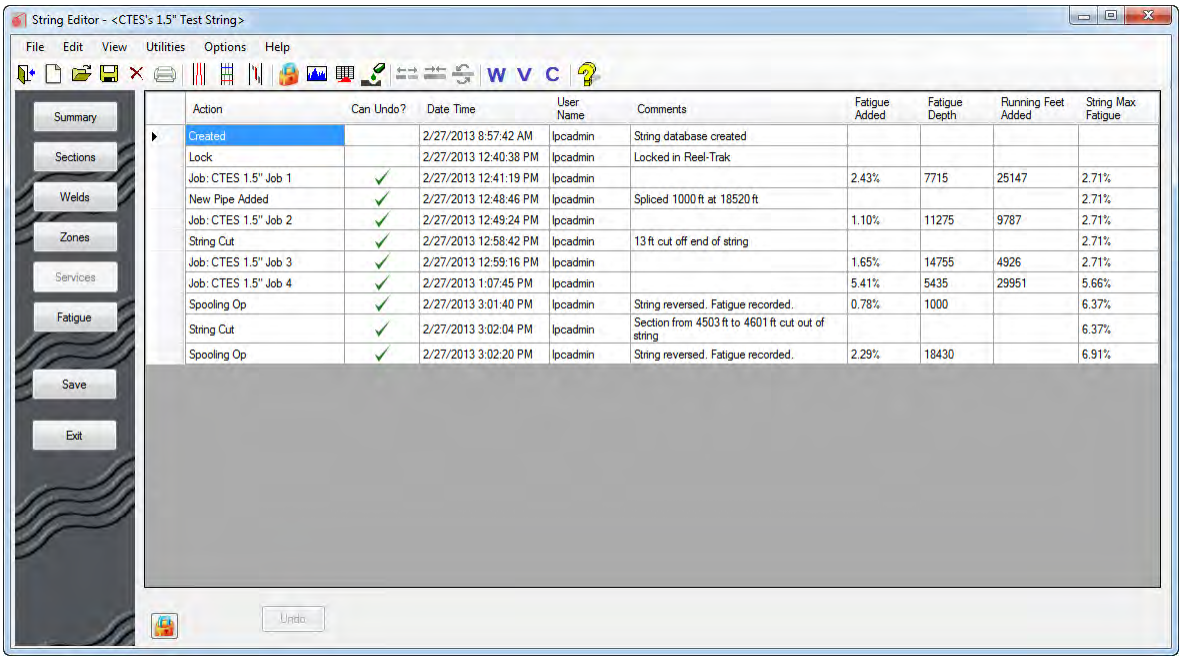
String history shows a history of the most important events that has happened to a string, and what jobs the string has performed. A user name is also associated with each event.

To open the String History display screen, click **History** on the toolbar or select **View > History Log** from the menu bar.

The columns **String Max Fatigue**, **Running Feet Added**, **Fatigue Depth**, and **Fatigue Added** store the data that was used in versions of Cerberus previous to v11.0 to generate comments after a job is executed.

➢ To use the column-related functionalities available in String History

1. Modify the comment for an action in the String History grid without losing the information recorded. The information stored in these columns is read-only.



Action	Can Undo?	Date Time	User Name	Comments	Fatigue Added	Fatigue Depth	Running Feet Added	String Max Fatigue
Created		2/27/2013 8:57:42 AM	lpcadmin	String database created				
Lock		2/27/2013 12:40:38 PM	lpcadmin	Locked in Reel-Trak				
Job: CTES 1.5" Job 1	✓	2/27/2013 12:41:19 PM	lpcadmin		2.43%	7715	25147	2.71%
New Pipe Added	✓	2/27/2013 12:48:46 PM	lpcadmin	Spliced 1000 ft at 18520 ft				2.71%
Job: CTES 1.5" Job 2	✓	2/27/2013 12:49:24 PM	lpcadmin		1.10%	11275	9787	2.71%
String Cut	✓	2/27/2013 12:58:42 PM	lpcadmin	13 ft cut off end of string				2.71%
Job: CTES 1.5" Job 3	✓	2/27/2013 12:59:16 PM	lpcadmin		1.65%	14755	4926	2.71%
Job: CTES 1.5" Job 4	✓	2/27/2013 1:07:45 PM	lpcadmin		5.41%	5435	29951	5.66%
Spooling Op	✓	2/27/2013 3:01:40 PM	lpcadmin	String reversed. Fatigue recorded.	0.78%	1000		6.37%
String Cut	✓	2/27/2013 3:02:04 PM	lpcadmin	Section from 4503 ft to 4601 ft cut out of string				6.37%
Spooling Op	✓	2/27/2013 3:02:20 PM	lpcadmin	String reversed. Fatigue recorded.	2.29%	18430		6.91%

2. Right-click the History grid to access the context-menu options:

 a. **Show/Hide Columns**.

 b. **Show Fatigue Profile**. Select this option to be redirected to the String History Timeline.

Undoing String Events

Events in a string's history can be undone. When a job is run, a string cut, or other events performed, Cerberus automatically makes a backup of the string before performing the action. When the user performs an 'undo' to the string, Cerberus retrieves the backup and replaces the current string with the backup.

String events that occurred in Cerberus versions earlier than 4.0 cannot be undone.

An "undo-able" row must be selected in order for the Undo option to be enabled.

A user cannot selectively undo an event in the middle of the string history. When the user performs an undo to an event, events that occur afterward will also be undone. Once a series of events has been undone, they cannot be

redone. If the user is unsure whether to perform the task, the string can be saved with a new name first.

If the user wishes to selectively remove an event from the middle of the string history, use the String Timeline instead of the history grid.

➢ To undo string events

1. Open String Editor.
2. Open the desired string.
3. Click **History**  on the toolbar.

 The string History shows a history of the most important events that have happened to a string, and what jobs the string has performed. A user is also associated with each event. There are rows for each event related to the string (creating, locking, unlocking, jobs run, cutting, reversing, and so forth). Events in red can be undone.
4. Select the desired event to undo.

 Select any column in the row for the event to undo; the **Undo** button is enabled.
5. Click the **Undo** button.

String Fatigue

Fatigue is viewed in the String Editor similarly to the way it is viewed in Reel-Trak (see “Reel-Trak” on page 29), but with the differences described below.

*If fatigue modeling capabilities are included in the user license, select **Edit > Fatigue Model** from the String Editor menu bar to specify these parameters.*

Most values in the Fatigue Model area cannot be changed if the string has fatigue.

➢ To view and use the string fatigue display

1. Complete one of the following:
 - Click the **Fatigue** icon  on the toolbar.
 - Select **View > Fatigue Profile** from the menu bar.
 - Select the **Fatigue** button in the left-hand portion of the String Editor display screen.
2. From the top right-hand **View** field, choose (from the dropdown) **Used Life** or **Diameter.**

a. When **Used Life** is chosen, the user sees:

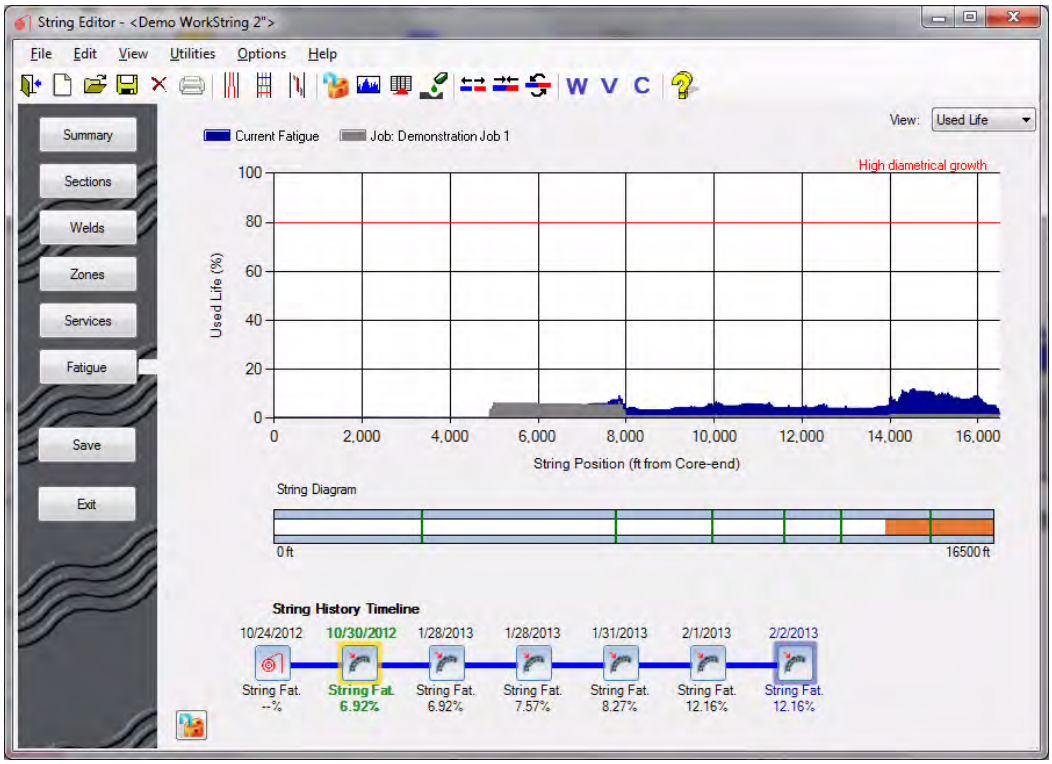
1. **Used Life (%)** diagram**.** In the diagram, plot orientation may be shown measured from the core end or from the free end of the string. To select either of these options, select **Options > Plot Orientation** and click (to check) the desired orientation.

2. **String Diagram -** This diagram shows the diameter, wall thickness, material, and welds on the string. Click close to a weld and hold the mouse button down to see the weld's position, type, and derating factor.

 The positions in the string diagram are measured from the core end of the string, as they were entered. The depth in the depth diagram comes from the depth information in the job log, and is unrelated to position.

The string diagram prints exactly as it appears on the screen.

3. **String History Timeline -** String Editor's fatigue diagram. This is a graphical representation of the string's history (without the lock and unlock actions).



Do not confuse the String History Timeline with the String History display. See String History for additional information.

➢ To use the String History Timeline

- If the string's backup file is available, hold the mouse over an event on the String History Timeline to display that action's incremental fatigue information in gray on the **Used Life** fatigue plot.

*The string's backup file is unavailable when it is imported from Cerberus v10.5 or previous (after-the-fact), or when it is created as a **Save As** file with the Save Criteria **Save Without Historical Data** selected.*

*The string's backup file is available when data is obtained directly in real time or when a **Save As** file is created with the **Historical Data** option selected. This backup file is created automatically in v11.0 or greater.*

- Any of the following user actions allow string history events to be changed in the specified manner to correct user errors without manually re-running the entire history. Fatigue is recalculated when one of the following actions is taken:

 a. Right-click an item on the **String History Timeline** and select **Insert Existing Job**, **Insert New Cut**, **Insert New Splice**, **Insert New Reverse**, **Remove**, or **Undo**.

 b. Right-click an item on the **String History Timeline** and select **Remove**. This action allows string history events to be deleted to correct user errors without manually re-running the entire history.

 c. Drag and drop an item on the **String History Timeline** to a new location on the timeline. This action allows string history events to be reordered to correct user errors without manually re-running the entire history.

 The **String History Timeline** checks the historic actions of the string to determine if items can be automatically recalculated if a change is made. If the item can be automatically recalculated, the line between items becomes **blue**. If items cannot be automatically recalculated, then the line becomes **grey**.

- The following items prevent automatic recalculation:

 a. String backup file missing

 b. Job database missing

 c. Reel database used on the job missing

 d. Splice with existing string

 e. Cut string and save as two separate strings

 f. The Cut, String or Reverse action does not have enough information to recreate

During the recalculation process (either manual or automatic), a dialog is displayed listing the actions that need to be recalculated. Manual recalculation requires the user to manually recreate each item in the list. **NOV CTES advises the user to save and/or print the list for reference**.



To . . .	Do this . . .
Zoom in on an area.	Click and drag a rectangle around the area.
Adjust the fatigue scale.	Right-click the diagram and select a scale.
Zoom back out.	Right-click the diagram and select **Zoom Out**.
Access additional graphic options.	Select **Options** in the menu bar; then choose from among the options available.

Troubleshooting in String Editor

Situation	Possible Cause	Action
I cannot make any changes to a string.	The string may be locked.	Try unlocking the string.

Situation	Possible Cause	Action
I can't change the segment length or fatigue model for a string.	The segment length or fatigue model for a string with fatigue cannot be changed.	Check to see if the string has any fatigue, or undo all history to remove the fatigue and the segment length can be changed.
I can't splice two strings together.	String Editor only allows strings with the same diameter, segment length, units, and fatigue model to be spliced.	Check to the diameter, segment length, units, and fatigue model for both strings.
I don't see a string in the list that I know should be there.	The string may be archived.	Try showing archived strings on the String Open dialog.
I get the error message "The string file doesn't exist." when I try to open a string.	The string database file may have been incorrectly deleted from the hard drive using File Manager or Windows Explorer.	This string cannot be recovered. Click **Yes** to remove the entry from String Editor.
The fatigue on the string seems very high.	If the string has performed many acid or sand cleanout jobs, the wall reduction may be unrealistically large. Since the wall thickness is so thin, fatigue accumulates on the string much more quickly. Also, if the string has performed high pressure jobs, the diameter may be increasing significantly, which also will accelerate the rate of fatigue accumulation.	Check the wall reduction for the string.

Chapter 11
Using the Tool String Editor

As the tool string configuration utility for Cerberus, the Tool String Editor provides the ability to create, edit, and manage tool strings. The Tool String Editor is used by Hydra, Orpheus, Solids Cleanout, and, optionally, CT Fatigue.

Starting the Tool String Editor

➢ To start the Tool String Editor

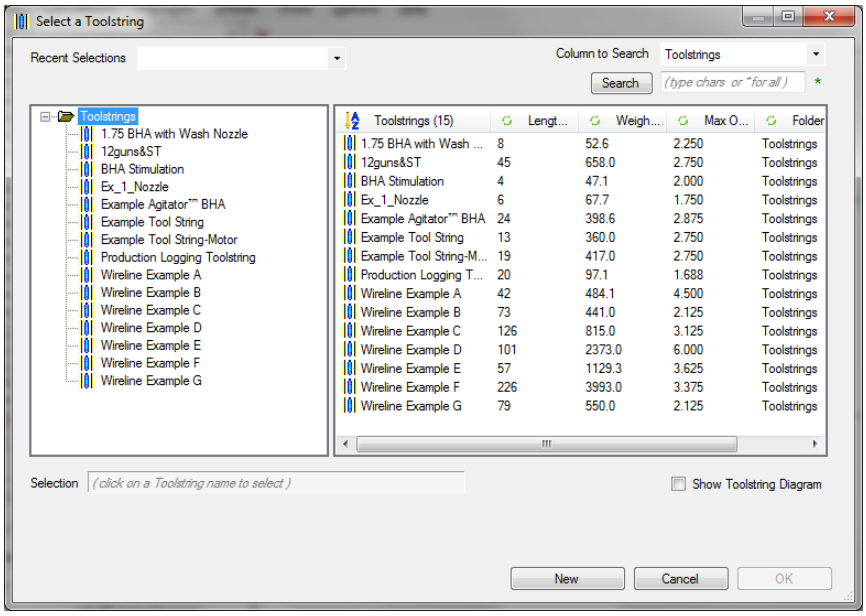
- Select the Managers menu-bar item
- Select Tool String Editor

OR

- Launch the Tool String Editor from any other Cerberus application where tool strings are used.

A screen similar to the following appears, from which one may select an existing tool string from the available list, or click **New** to create a new tool string.

Select a Tool String

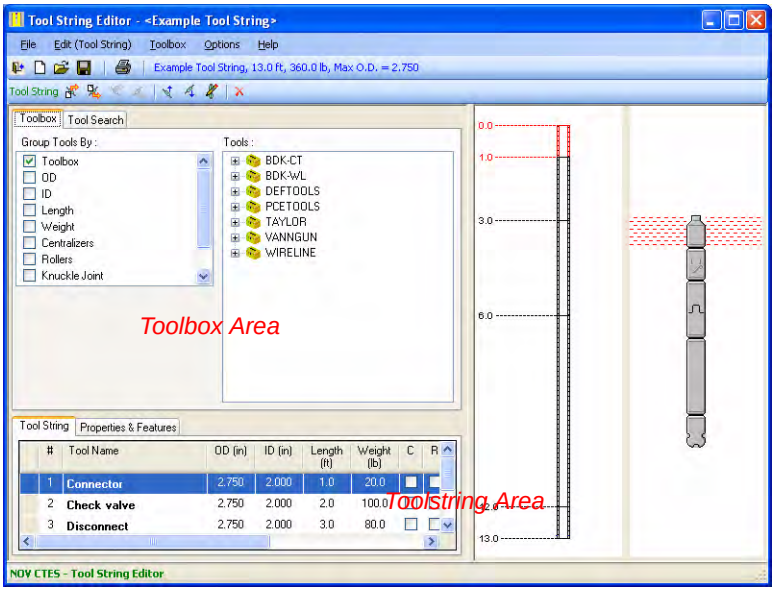


Navigating the Tool String Editor

The Tool String Editor window is divided into two primary functional areas:

- Toolbox
- Toolstring

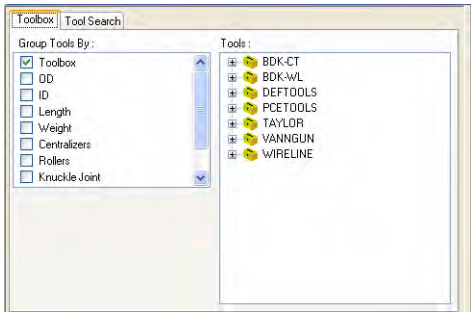
Tool String Editor Window



Toolbox Area

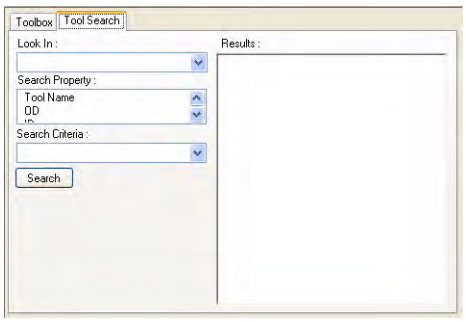
The Toolbox Area contains two tabs, the Toolbox tab and the Tool Search tab, that are useful for organizing or searching for tools, creating and managing toolboxes, and selecting tools to insert into the tool string.

Toolbox Tab



The Toolbox tab is the access point for creating tools, for organizing toolboxes, and for grouping tools with the purpose of locating and inserting them into a tool string.

Tool Search Tab

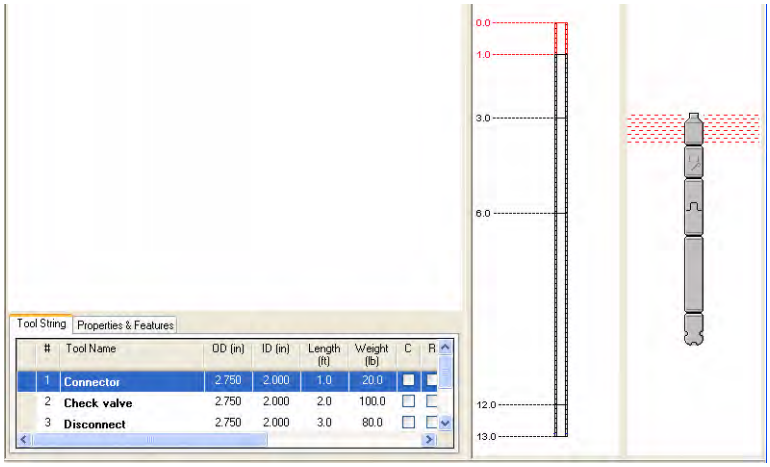


Similar to the Toolbox tab, the Tool Search tab is used to search for, to locate, and to insert a particular tool into a tool string.

ToolString Area

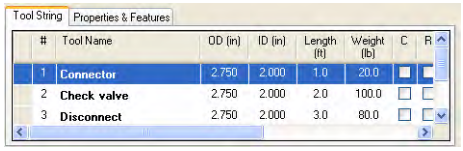
The Tool String Area, comprised of the display-screen area not previously described, is used to edit the tool string, to display tool properties, to edit tools within the tool string, and to render a schematic and picture of the tool string.

Tool String Area



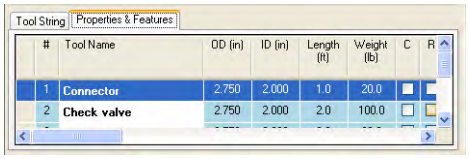
The Tool String tab shows a selection of properties related to the tools included in the current tool string.

Tool String Tab



The Properties & Features tab shows all of the tool properties.

Properties & Features Tab



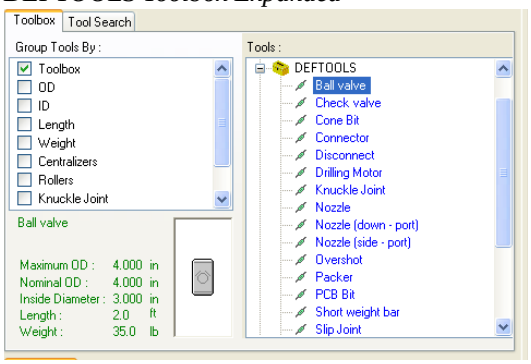
Within the Cerberus Tool String Editor tool bar, the tool string's total length, total weight, and maximum O.D. are displayed.

Working with Toolboxes

Toolboxes contain individual tools that, once defined and saved, can be inserted into many different tool strings. Toolboxes can be created, and tools can be copied to, and removed from, any toolbox.

By default, tools are grouped only by the toolbox in which they are located. In the figure below, the DEFTOOLS toolbox is expanded to list the individual tools it contains, and the Ball valve tool is selected. A subset of the properties pertaining to the selected tool, as well as a picture associated with the tool, are displayed in the bottom left portion of the Toolbox tab.

DEFTOOLS Toolbox Expanded



Creating a New Toolbox

➢ To create a new toolbox

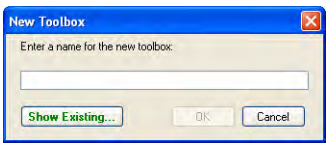
1. Select **Toolbox > Create New Toolbox.**

 OR

 From the Toolbox tab, right-click in the Tools area and select **Create New Toolbox**.

2. The New Toolbox dialog appears.

New Toolbox Dialog



3. Enter a toolbox name. The name must not be that of an existing toolbox and must meet standard Windows file-naming criteria.

4. The new toolbox is added to the Tools area inside the toolbox in which it was created.

Creating a New Tool Inside a Toolbox

Because many different tool strings can use the same tool, it is good practice to store a much-used tool in an easily accessible toolbox for later string insertion.

➢ To create a new tool inside a toolbox

1. Select an existing toolbox within the Tools area, or create a new toolbox (see To create a new toolbox for more information).

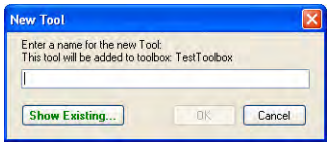
2. Select **Toolbox > New Tool.**

 OR

 From the Tools area, right-click the intended toolbox and select **New Tool**.

3. The New Tool dialog appears, within which is included the name of the toolbox where the tool is to be created.

New Tool Dialog



4. Enter a tool name. The name must not match that of an existing tool.

5. The Tool Editor display screen appears, where general tool information, tool properties, and tool connections pertaining to the new tool may be input.

6. Click **Save** (lower left-hand portion of the display screen) to save the tool with the name specified at creation, or click **Save As...** to save the tool with another name.

7. The new tool is saved, added to the Tools area inside the first-specified toolbox, and is available for selection.

Editing a Tool Within a Toolbox

Use the Tool Editor to edit tools residing in a toolbox.

➢ To launch the Tool Editor and edit an existing tool

1. Select the tool in the Tools area.

2. Select **Toolbox > Edit Tool.**

OR

1. Right-click the tool in the Tools area.

2. Select **Edit Tool.**

3. Edit general tool information, tool properties, and tool connections pertaining to the tool.

4. Click **Save** (lower left-hand portion of the display screen) to save the tool with the name specified at creation, or click **Save As...** to save the tool with another name.

After saving the tool, the Tool Editor asks whether or not the edit(s) should be applied to any tool string containing the tool.

Deleting a Tool from a Toolbox

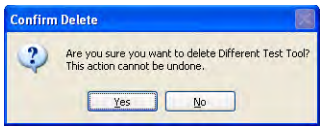
➢ To delete a tool from a toolbox

1. Select the tool in the Tools area.
2. Select **Toolbox > Delete Tool.**

OR

1. Right-click the tool in the Tools area.
2. Select **Delete Tool.**
3. A confirmation dialog appears, in which it is noted that deleting a tool cannot be undone.

Confirm Delete Dialog



Copying an Existing Tool Into a Tool Box

Copy tools from one toolbox to another by dragging them in a manner typical of Windows-based applications.

➢ To drag/copy a tool into a toolbox

1. Click the tool name in the Tools area
2. Drag the tool by holding the left mouse button down and moving the mouse
3. Release the mouse when it hovers over the target toolbox

Similarly, drag/copy tools from a tool string into a tool box:

1. Click the tool name in the Tool String tab, or click the tool in the Tool String schematic
2. Drag the tool into a toolbox within the Tools area

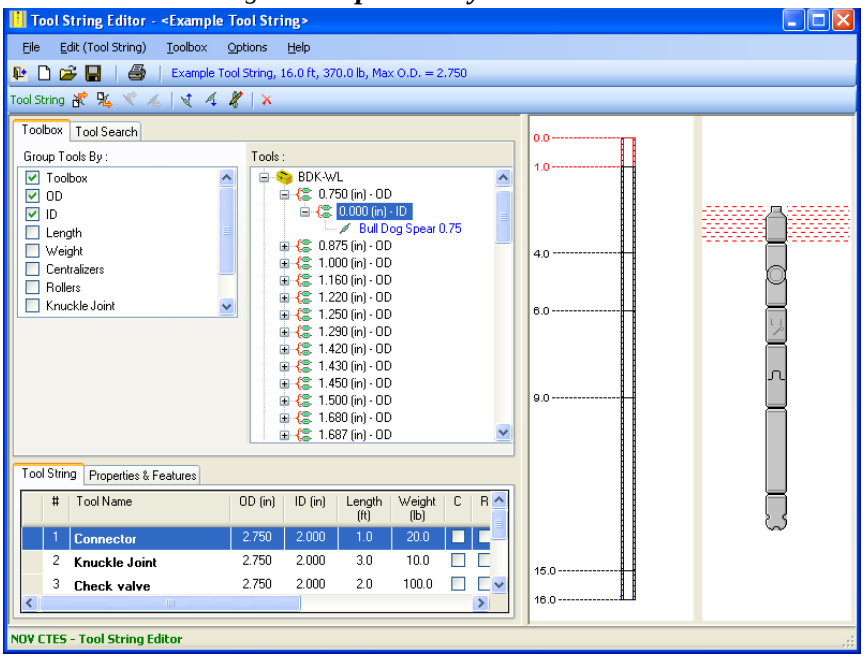
Finding Tools by Grouping

In previous version of Cerberus, filters could be applied to the selected toolbox to limit the number of tools displayed. In the Tool String Editor, rather than filtering, tool selections can be refined by grouping tools based on their properties.

Tools, by default, are grouped within the Tools area according to toolbox; they can be grouped differently, as follows:

- Check or uncheck tools within the Group Tools By area. Items in the Tools area become sorted according to the checked Group Tools By items.
- Drag items in the Group Tools By area up or down to manually sort them.

***Tools** Sorted According to **Group Tools By** Checkboxes*



Tool-grouping order is "remembered" between application sessions.

➢ To add items to the Group Tools By area

1. Right-click an item in the Tool String tab (within the Tool String area)
2. Click User Selected Columns
3. Select a column that is not already checked

The column appears in the Group Tools By area.

Finding Tools using Tool Search

➢ To use the Tool Search tab to find specific tools

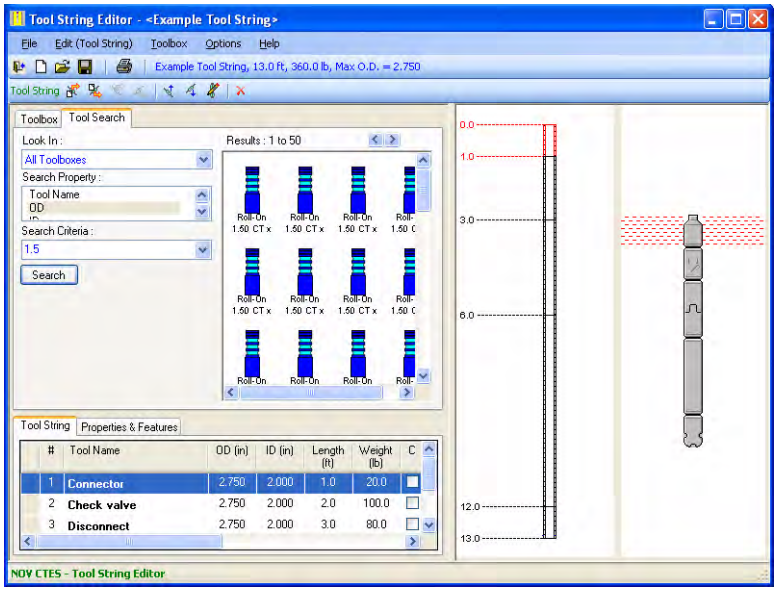
1. Select the Tool Search tab
2. Using the Look In dropdown, select a search location
3. Select a Search Property

To add Tool String area items to the Search Property selection field, right-click anywhere in the Tool String tab, click User Selected Columns, and select the desired column.

4. Enter the Search Criteria

5. Click **Search**

For example, to search in All Toolboxes for tools that have a 1.5 inch OD, the following values are entered, and the following results produced:

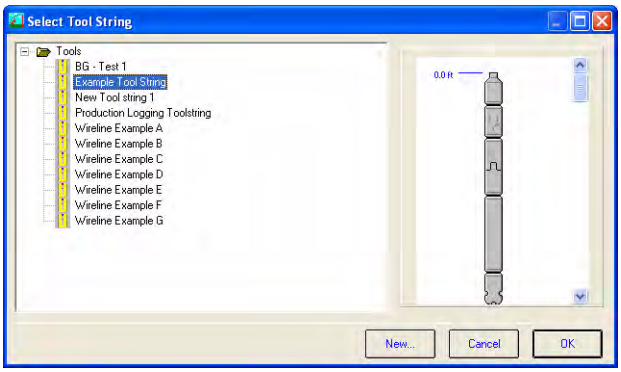


Working with Tool Strings

Create, modify, delete, import, and export tools strings using the Tool String Editor.

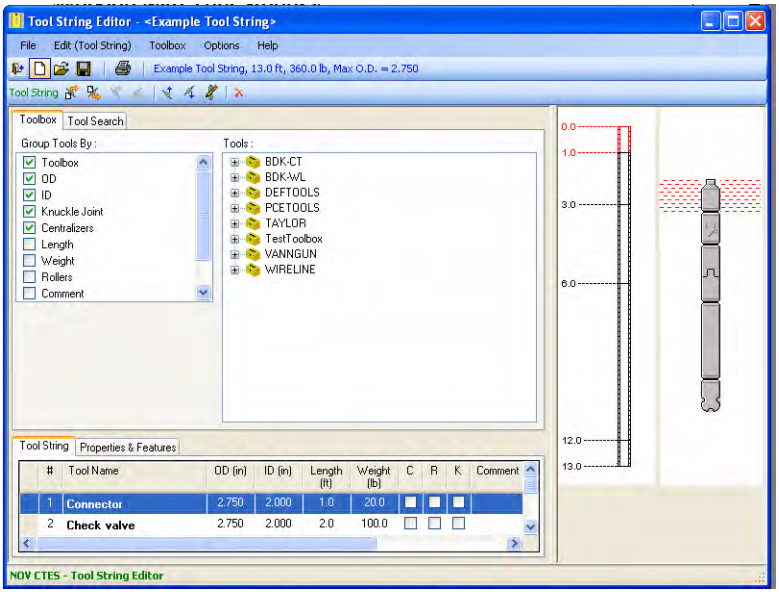
➢ To create a new tool string

1. Use one of the following methods to access the New Tool String dialog:
 - From the Select Tool String display screen, click **New**.

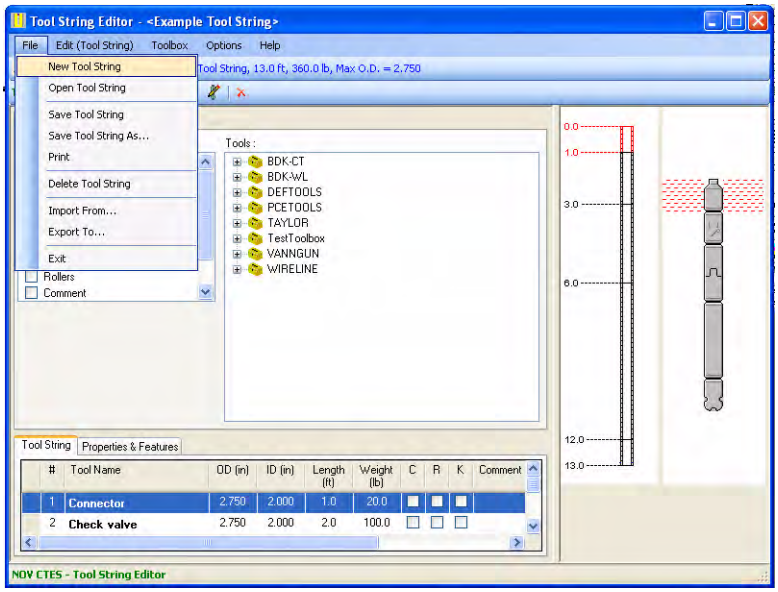


*The Select Tool String display screen appears immediately after launching the Tool String Editor from the **Managers** menu-bar item in Cerberus.*

- From within the Tool String Editor, click the **Create a new tool string** button in the toolbar.



- From within the Tool String Editor, select **File > New Tool String**.

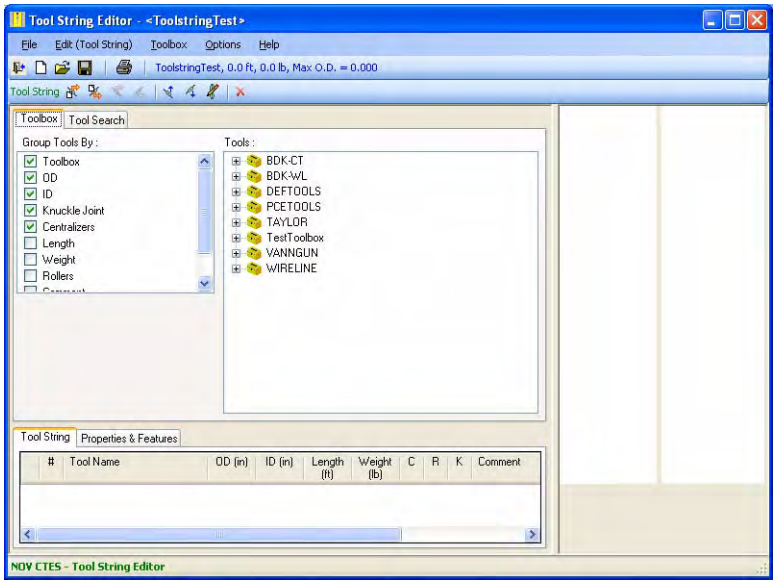


The New Tool String dialog appears:



2. Enter a name for the new toolstring and click **OK**.

An empty tool string appears:



Adding Tools to a Tool String

Insert an existing tool from a toolbox or create a new tool.

➢ To insert a tool from a toolbox

1. Locate the tool by grouping or by executing a tool search (see Finding Tools by Grouping or Finding Tools using Tool Search for more information).

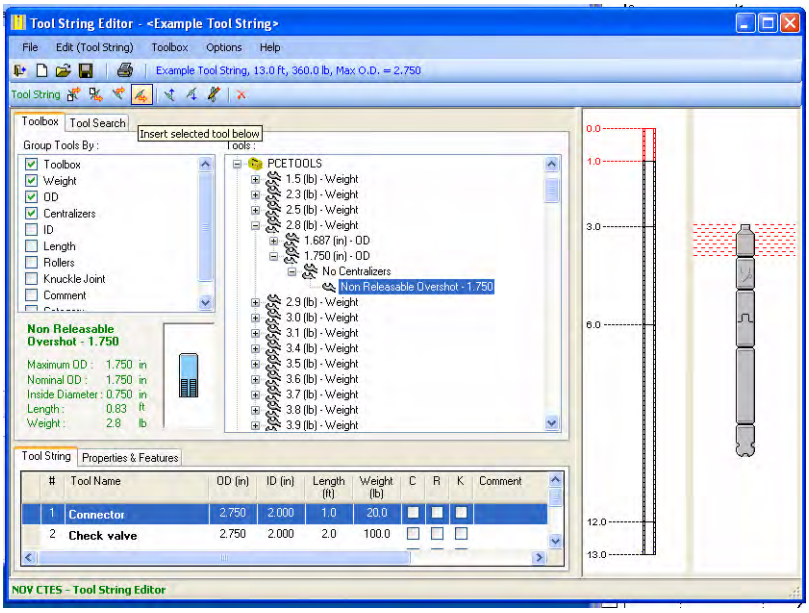
2. Click to select a Tool Name in the Tool String tab.

The Tool Name selected in the Tool String tab affects the to-be-inserted tool's position in the hierarchy. See Step 4. below for more information.

3. Click to select the located tool in the Tools area.

4. Insert the tool using **one** of the following methods:

 - Select **Edit (Tool String)** in the menu bar; click **Insert from Toolbox Below** to position the inserted tool below the currently selected tool in the Tool String, or click **Insert from Toolbox Above** to position the inserted tool above the currently selected tool in the Tool String.
 - Click the toolbar item **Insert selected tool below** or click the toolbar item **Insert selected tool above.**



- Right-click the desired tool; click **Insert from Toolbox Below** or **Insert from Toolbox Above**.
- Drag the tool to the schematic area or to the Tool String tab.

When tools are added, the total length, weight, and Max O.D. for the tool string are updated in the tool bar. The newly added tool is rendered in red in the schematic, and shaded red in the picture.

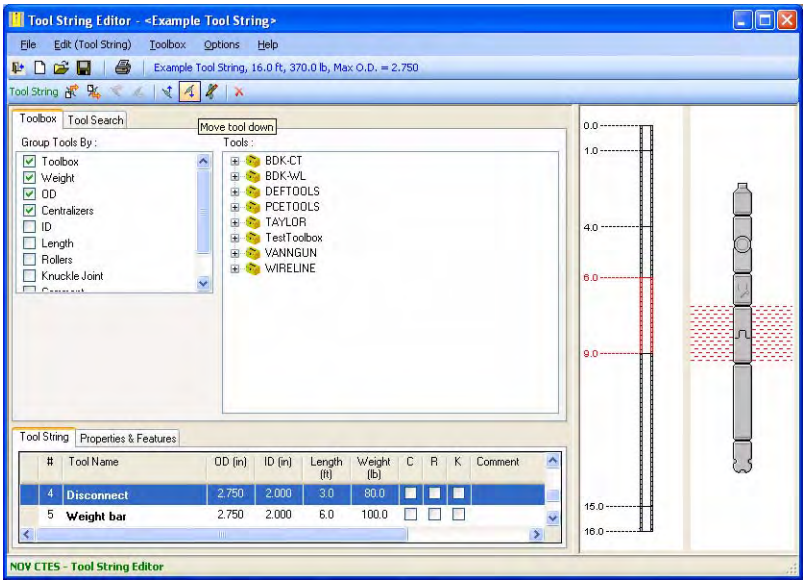
Moving Tools within a Tool String

Move tools up or down in a tool string relative to other tools in the string.

➢ To move tools within a tool string

1. Select the tool in the schematic or in the Tool String tab.
2. Move the tool using one of the following methods:
 - Select the **Edit (Tool String)** menu-bar item; click **Move Tool Up** or **Move Tool Down.**

- Click the **Move Tool Up** or **Move Tool Down** toolbar-menu item.



- Right-click the tool on the schematic or on the Tool String tab; click **Move Tool Up** or **Move Tool Down.**
- Drag the tool up or down the string.

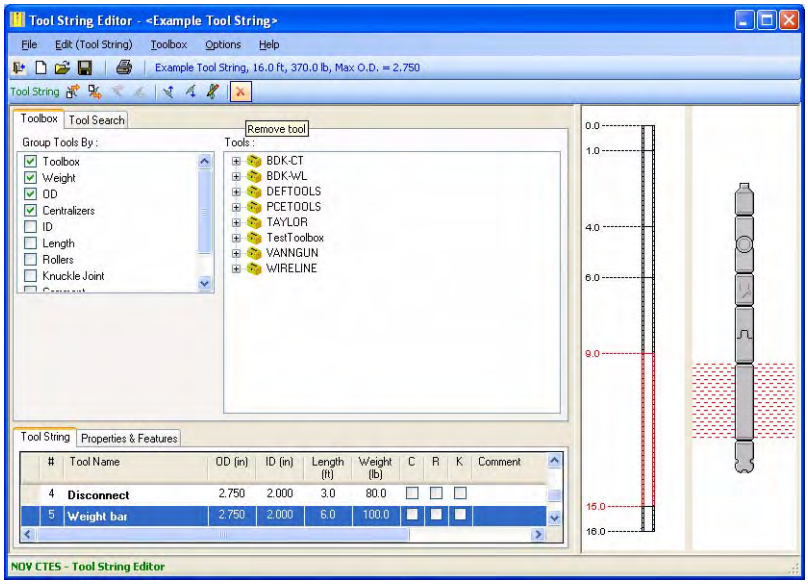
To drag a tool up or down the string, click to select it in the schematic or in the Tool String tab, and then drag it by holding the left mouse button depressed. When the blue arrow in the schematic (or the shaded line in the Tool String tab) reaches the desired location for the tool, release the mouse button to drop the tool in the new location.

Removing Tools from a Tool String

➢ To remove a tool from a tool string

1. Select the tool in the schematic or in the Tool String tab.
2. Remove the tool using one of the following methods:
 - Select the **Edit (Tool String)** menu-bar item; click **Remove Tool.**

- Click the **Remove Tool** toolbar-menu item.



- Right-click the tool on the schematic or on the Tool String tab; click **Remove Tool.**

Removing a tool from a tool string does not delete it from any toolbox.

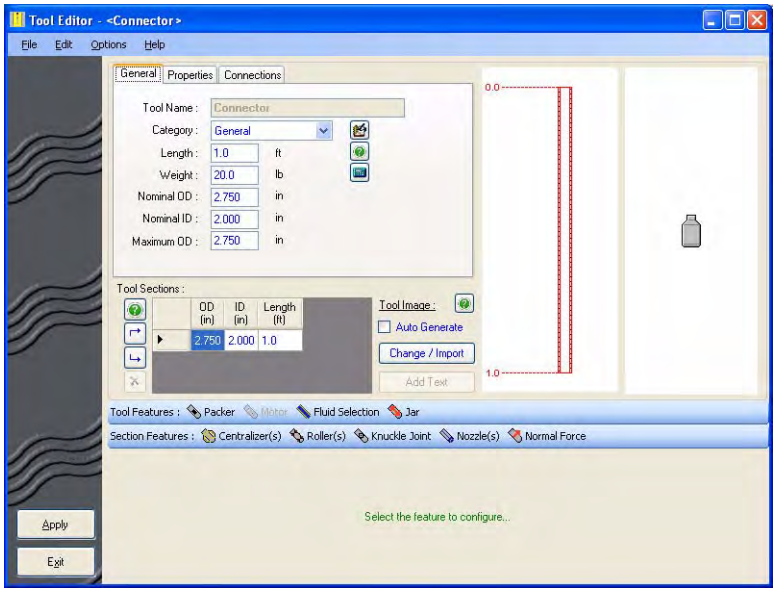
Editing a Tool within a Tool String

Edit a tool, independently of any toolbox, inside the tool string using the Tool Editor.

➢ To edit a tool within a tool string

1. Select the tool in the schematic or in the Tool String tab.
2. Launch the Tool Editor using one of the following methods:
 - Select the **Edit (Tool String)** menu-bar item; click **Edit Tool.**
 - Right-click the tool on the schematic or on the Tool String tab; click **Edit Tool.**
 - Double-click the tool in the Tool String tab or on the schematic.

The Tool Editor appears.



Editing a tool inside a tool string does not update it in any toolbox. Also, a tool is not saved inside an edited tool string unless the tool string is saved.

Working with the Tool String Tab

The Tool String tab, like the Properties & Features tab, displays a tool string as a table of data, with a subset of tool properties and features.

Data displayed in the Tool String tab is configurable so that only properties or features of particular interest are displayed on the main tab.

➢ To configure the Tool String tab

1. Right-click anywhere on the Tool String table and move the mouse to the User Selected Columns item.

 A submenu appears, listing all the available columns, with the selected columns checked.

2. Toggle (check or un-check) an item on this list to show or hide the column in the Tool String tab.

 Items checked in this list are also available in the Group Tools By list on the Toolbox tab and the Search Property list on the Tool Search tab.

The Tool Name and # fields are not available for selection; they are always visible.

3. Resize the table columns by using the mouse to drag the border between two column headings.

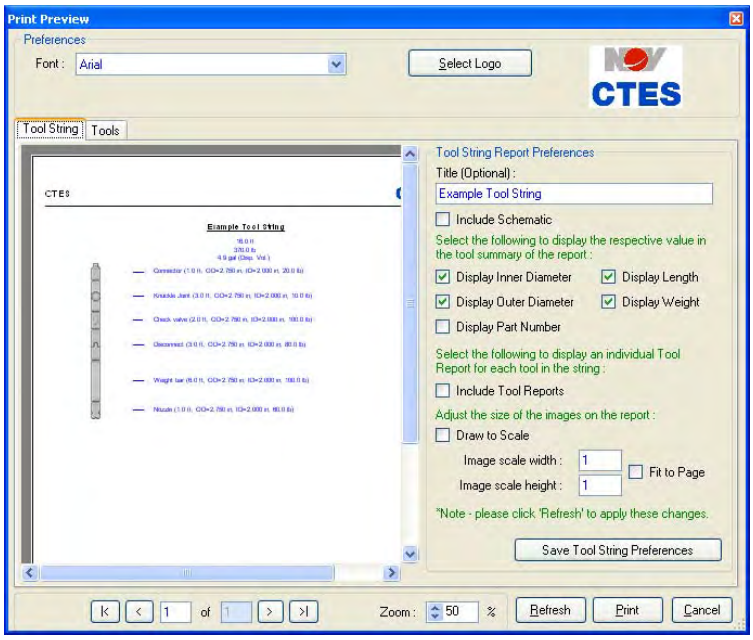
The Tool String tab's configuration is "remembered" between sessions.

Printing a Tool String

➢ To print a Tool String

1. From the Tool String Editor display screen, select **File > Print**.

 The Print Preview display screen appears



2. Review and customize the to-be-printed Tool String.

 a. The Tool String tab shows a print preview of the entire tool string.

 b. The Tools tab contains a collection of individual printouts for each tool in the string.

Check the Include Tool Reports togglebox to include individual tool reports in the printout.



After customization, save the settings in the Tool String Preferences panel for the next session by clicking the same-named togglebox.

Chapter 12
Using the Tool Editor

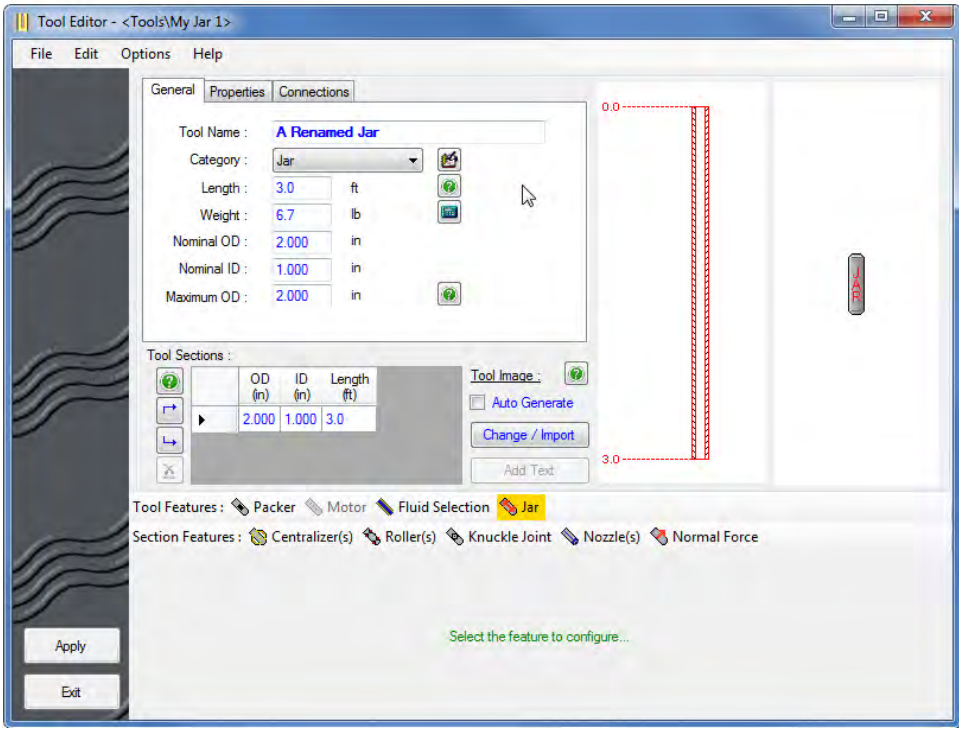
The Tool Editor, launch-able from the Tool String Editor, is used to edit tools inside a toolbox or within an individual tool string.

About the Tool Editor

The Tool Editor display screen contains a set of tabs that detail the properties of the tool, a panel to display a schematic rendering of the tool, a panel to display an image of the tool, a table listing tool sections (described later), and toolbars for accessing and modifying the features of the tool and the individual tool sections.

Launching the Tool Editor

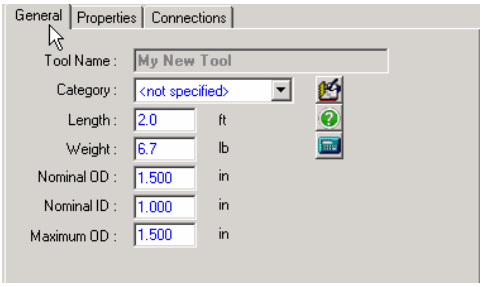
Launch the Tool Editor by double-clicking on a tool name in the Tool String Editor (see Using the Tool String Editor for more information). A display screen similar to the following appears:



Setting Tool Properties

Define the general properties of the tool using the collection of tabs at the top of the screen.

The **General** tab contains standard tool properties, such as **Tool Name**, **Category**, and physical dimensions, all of which are required, with the exception of **Category**.



- The **Tool Name** cannot be modified. Use **File > Save As** to save the tool under a new name.

- Edit the available **Categories**:

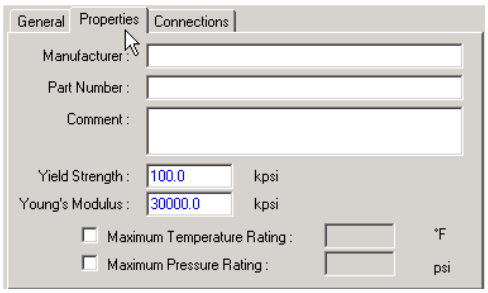
 Click the button to the right of the **Category** field.

 OR

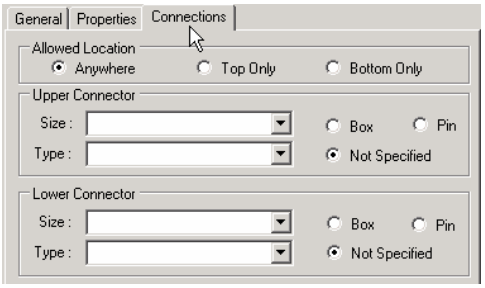
 Select **Edit > Toolbox > Categories** from the menu bar.

- Estimate the **Weight**, based on tool dimensions, using the calculator button to the right of the **Weight** field.

The **Properties** tab contains additional tool properties that can be defined if desired, including a **Manufacturer**, **Part Number**, **Comment**, **Maximum Temperature** or **Pressure Rating**, **Yield Strength**, and **Young's Modulus**.



Use the **Connections** tab to specify the type of connector for the tool, as well as the allowable location for the tool within a tool string.

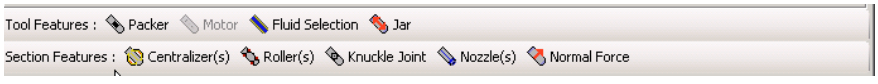


*Only the **Allowed Location** is validated by the Tool String Editor when adding tools to a tool string; the connector types and sizes are not validated.*

Setting Tool Features

Tool features are more specific characteristics a tool has beyond its set of physical properties. Tool features can include details such as whether or not the tool is a packer, a drilling motor, or a jar.

Configure tool features by opening each feature panel from the Tool Features or Section Features toolbar at the bottom of the Tool Editor display screen.



When tool features have been defined, the specific feature appears in bold text in the toolbar.

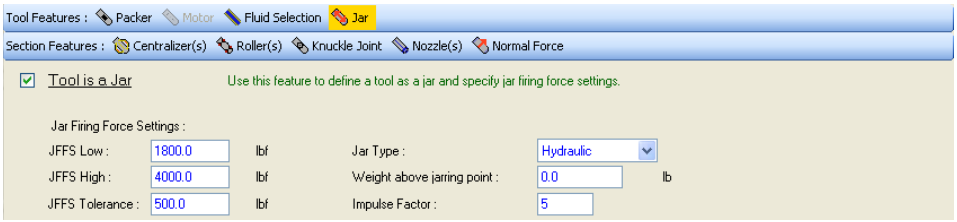
Tool Features and Section Features are listed separately because the Tool Editor can be used to define multiple sections on a single tool. This is covered later in this section, but this capability allows the location of features like nozzles, rollers, knuckle joints, and centralizers to be more specifically located on the tool. By default, each tool has only one tool section.

Each feature panel provides all the data entry fields needed to define the feature for use in Cerberus calculation engines.

Setting Advanced Tool and Section Features

Tool Features and Section Features requiring additional discussion are described here.

Setting Jar Tool Features (WL)



➢ To set jar tool features

1. Within the Tool Editor, specify that the tool is a jar.
 a. Select the **Jar** Tool Feature.
 b. Click to check the togglebox associated with the **Tool is a Jar** field.
2. Set the **Jar Firing Force Settings (JFFS)**.

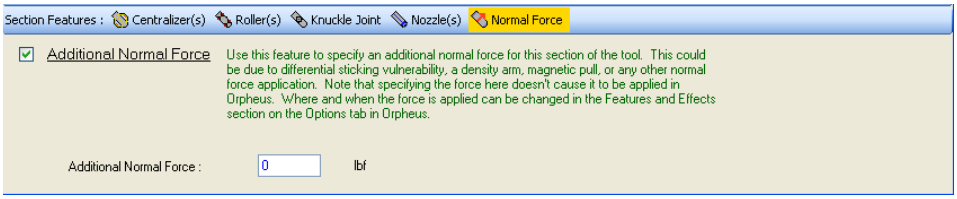
Jar firing force is the force at the jarring point required to fire the jar. If the force at the jarring point is within a firing-force range (between **JFFS Low** *and* **JFFS High***), the jar fires. The jar also fires due to force at jarring of* **JFFS Tolerance** *lower than* **JFFS Low** *or higher than* **JFFS High***.*

Using a Jar Tool in Calculations

Surface weight and cablehead tension corresponding to low- and high-setting (respectively) jar firing forces may be calculated in Orpheus.

See Completing Jar Activation Calculations (WL) for more information concerning using a jar tool in calculations.

Setting Normal Force Section Features (WL)



Use the Normal Force feature to specify an additional normal force for any tool section. This force could be used to represent a vulnerability to differential sticking, a density arm, magnetic pull, or some other force the modeler would like to see reflected in Orpheus calculations.

➢ To set Additional Normal Force

1. Within the Tool Editor, specify that additional normal force is a tool-section feature.
 a. Select the **Normal Force** Section Feature.
 b. Click to check the togglebox associated with the **Additional Normal Force** field.
2. Input the **Additional Normal Force**.

Using Additional Normal Force in Calculations

Specifying additional normal force in the Tool Editor doesn't cause the force to be applied in Orpheus. The force's application can be included or not included in the Features and Effects section underneath the Options tab in Orpheus. See Including Additional Normal Force in Calculations (WL) for additional information.

Setting the Tool Image

Change the image associated with a specific tool using the Tool Editor. By default, the tool image is a very simple image including a question mark. The tool image can be changed to any image available on disk or any existing image in another toolbox. Additionally, the Tool Editor can attempt to render an image that represents the tool based on the physical dimensions, features, and properties defined for the tool.

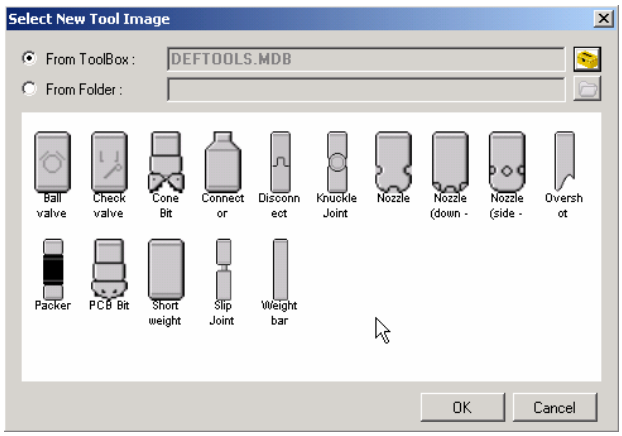
➢ To select a different image for the tool

Click the **Change / Import** button under the **Tool Image** heading in the middle of the screen.

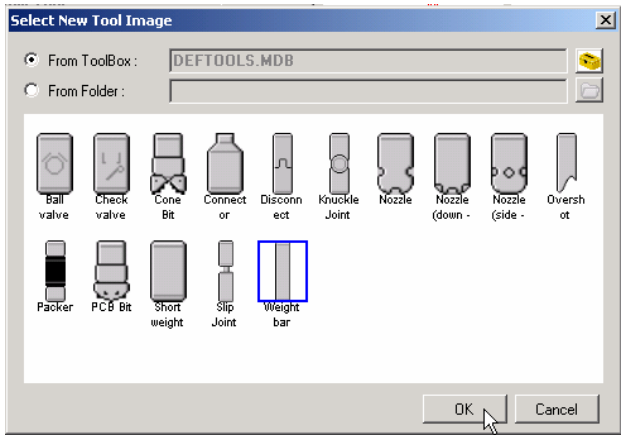


After clicking the **Change / Import** button, the Select New Tool Image display screen loads. Here, select an image from an existing toolbox or from any folder on a disk accessible by the computer.

In the example below, the **DEFTOOLS** toolbox has been selected as a source for images:



To change the image associated with the tool, select an image from the dialog and click **OK**.



In addition to choosing the image manually, the Tool Editor can attempt to render an image based on the tool properties and features.

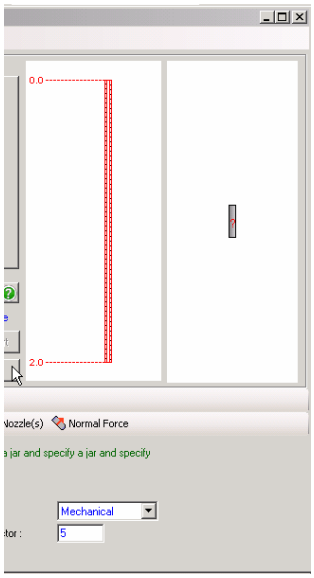
➢ To render an image based on tool properties and features

Click (to check) the **Auto Generate** box under the **Tool Image** heading in the middle of the dialog.

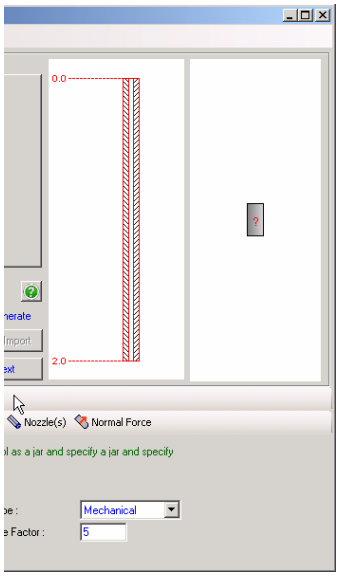


While this box is checked, changing certain tool properties and features results in a change to the tool image. In the example below, the **Nominal OD** is increased and then a centralizer is added.

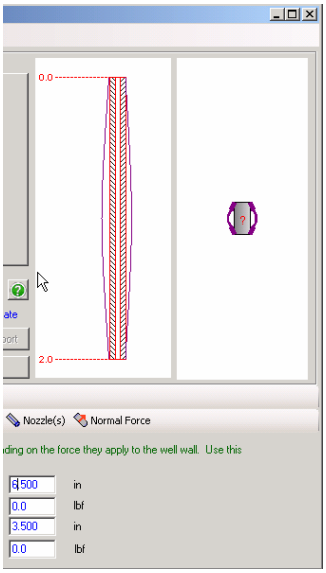
Original



Nominal OD Increased



Centralizer Added

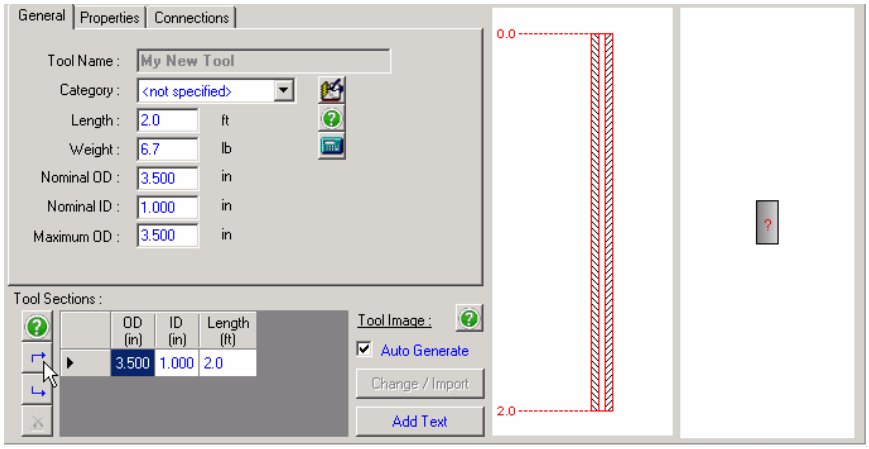


Defining Tool Sections

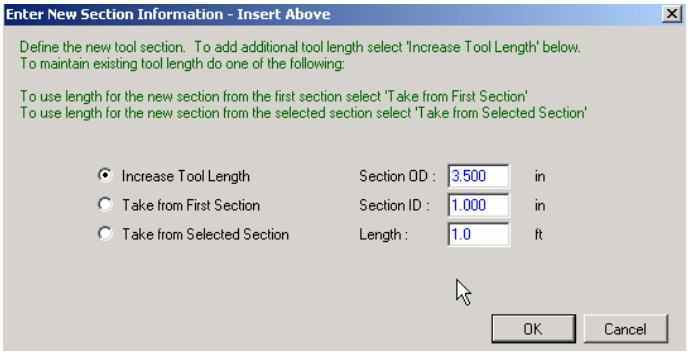
The Tool Editor can be used to define multiple sections on a single tool. This capability allows the location of features like nozzles, rollers, knuckle joints, and centralizers to be more specifically defined for the tool. By default, each tool has only one tool section and only one is needed for Cerberus calculation engines.

To add a section to a tool

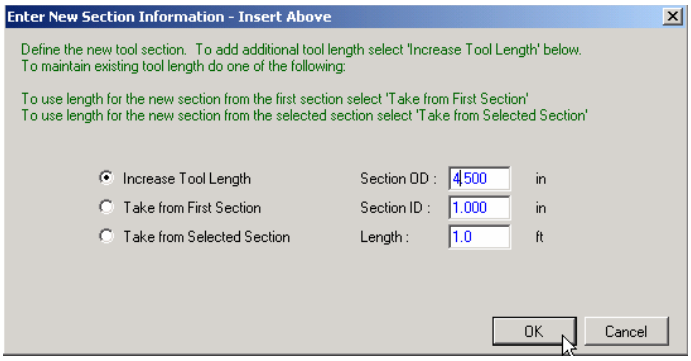
1. Click the **Insert Section Above** or **Insert Section Below** button under the Tool Sections heading in the middle of the screen.



Either button launches the Enter New Section Information dialog, in this case inserting a section above:

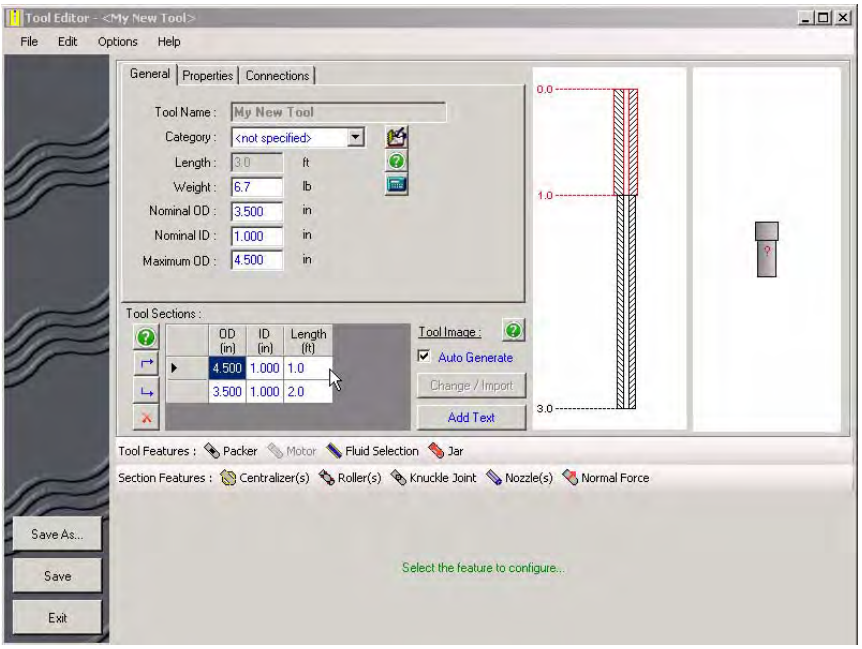


2. Select **Increase Tool Length**, **Take** length **from** the **First** tool **Section** of the tool, or **Take** length **from** the currently **Selected Section** of the tool. In the example below, the Increase Tool Length option is selected and the section OD is changed to 4.5 inches:



3. Click **OK** in the Enter New Section Information dialog.

The new section is inserted above the existing section with the OD, ID, and length provided. Additional sections may be entered, if desired.



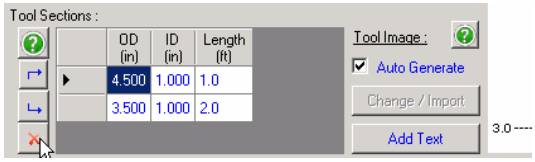
When more than one tool section exists on a tool, the currently selected section is highlighted in the Tool Sections table and rendered in red in the schematic view on the right side of the screen.

Because the Auto Generate option is selected, the tool image is updated to reflect the additional tool section.

The Length field is disabled on the General tab when multiple tool sections are defined because the length of the tool is now calculated as the sum of the individual tool sections.

➢ To delete a tool section

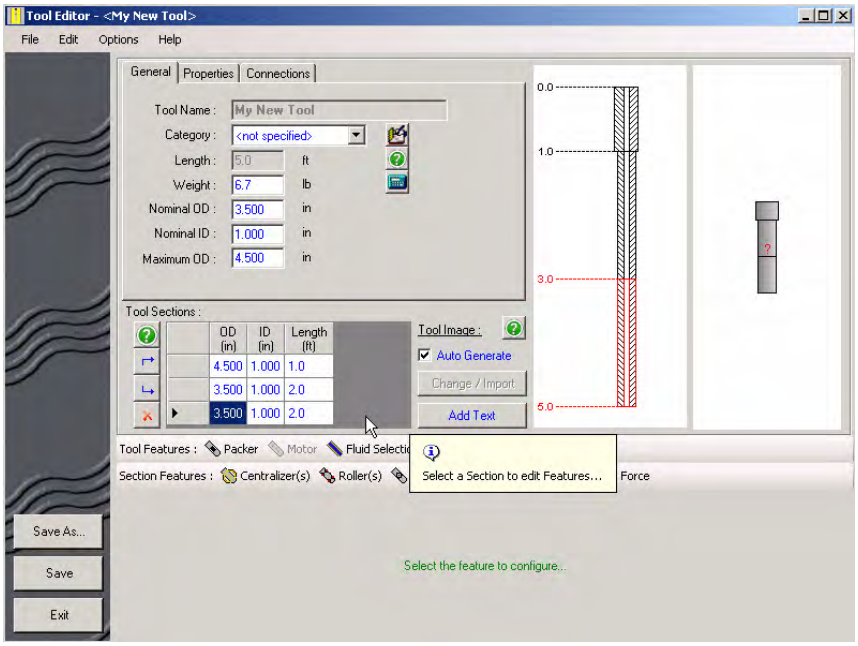
Click the **Delete Section** button under the Tool Sections heading in the middle of the dialog.



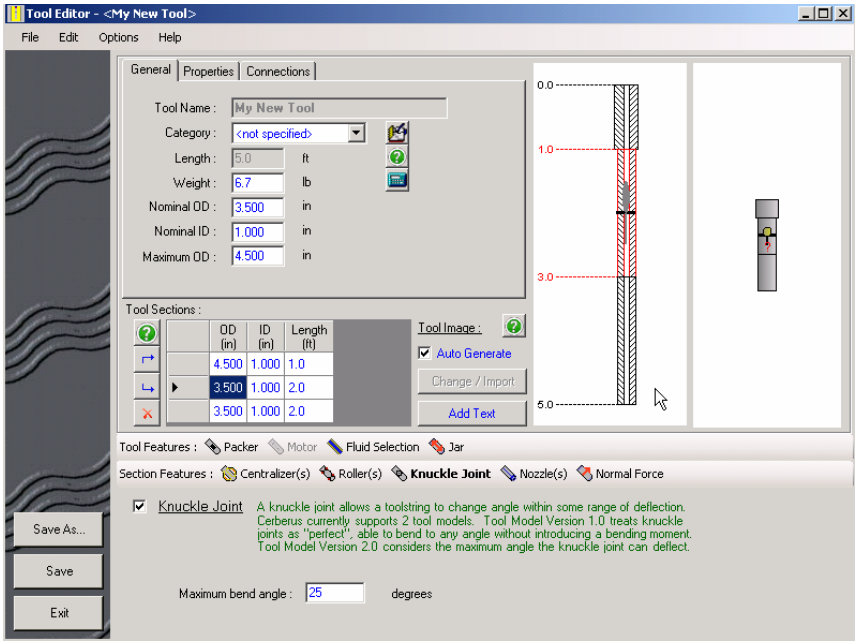
Defining Tool Section Features

By default, each tool has only one tool section. Additional tool sections are not required by Cerberus calculation engines. Once defined, however, a set of section features exists for each tool section. This provides the capability to define a tool with multiple centralizers, rollers, knuckle joints, nozzles, or specific areas where a normal force is applied.

In the example below, a tool has been defined with three tool sections:

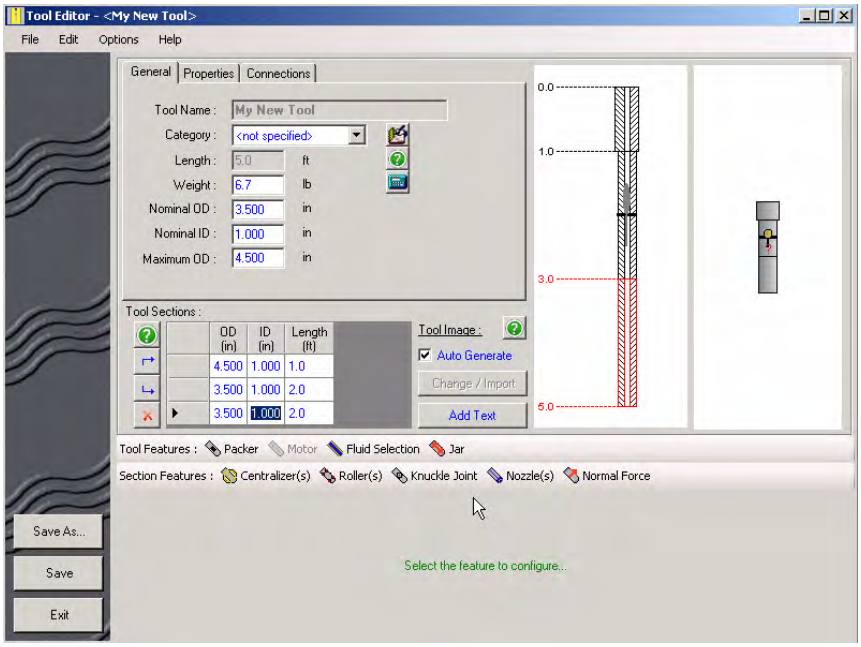


After selecting the middle section, a Knuckle Joint feature is added with a maximum bend angle of 25 degrees:

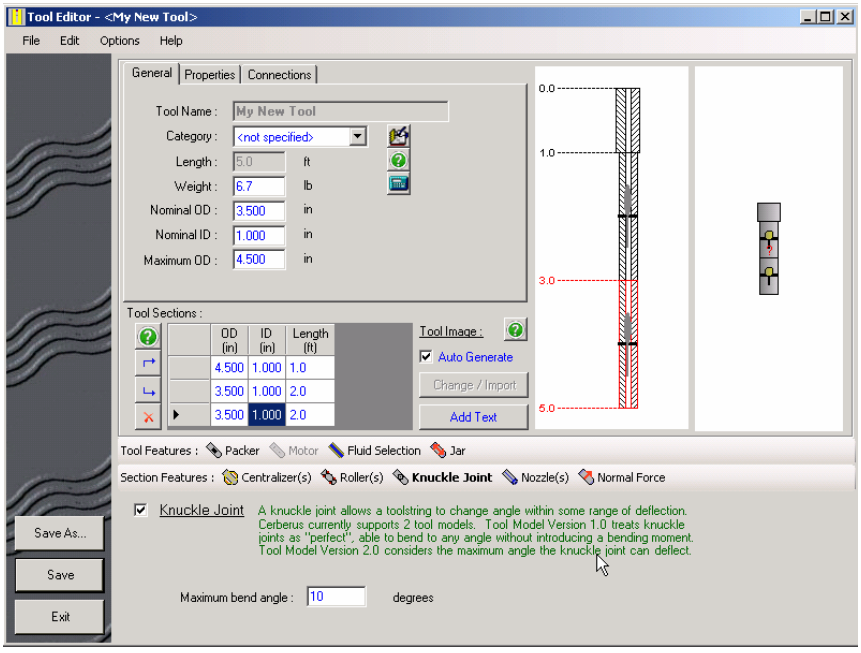


The Knuckle Joint Section Feature is now bold, the schematic has been updated to reflect the knuckle joint on the middle section, and the tool image has been updated to reflect the knuckle joint because Auto Generate is selected.

In the following image, the last section of the tool is selected. Notice that the Knuckle Joint section feature is not bold. This is because the knuckle joint is defined for the middle section of the tool:



This means an additional knuckle joint can be defined on the same tool, in the top or bottom section. In the case below, an additional knuckle joint is defined on the bottom section with a maximum bend angle of 10 degrees:



The Knuckle Joint Section Feature is now bold for both tool sections that have a knuckle joint defined; the schematic has been updated to reflect that there is a knuckle joint defined for the middle and bottom section of the tool; the tool image has been updated to reflect the knuckle joint because Auto Generate is selected.

Printing a Tool

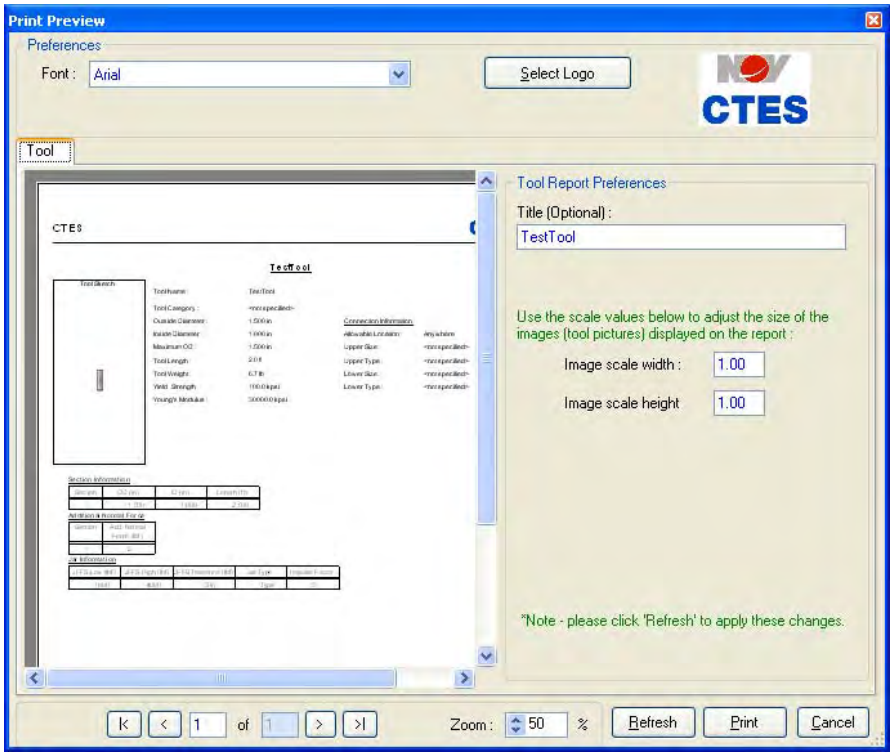
➢ To print a Tool

1. From the menu bar of the Tool Editor display screen, select **File > Print**.

 This launches the Print Preview screen, which displays the printout and allows for customization and review prior to printing.

2. From the Print Preview screen the tool can be printed by clicking the **Print** button.

In the example below, the report has been zoomed in. Notice that the section information is included on the report as well and the knuckle joints are defined by each section.



Customizing a Tool Printout

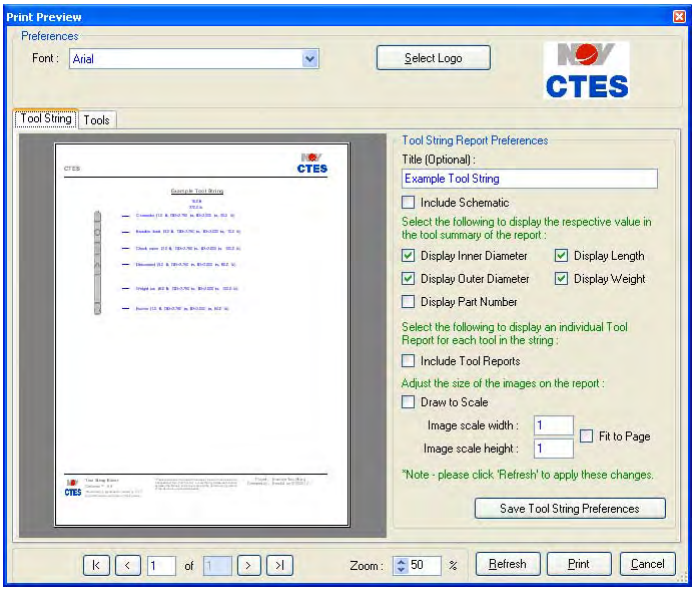
There are several options that are selectable to customize the tool printout – including selecting a different logo for the header of the printout, selecting a different font, altering the scale used to render the tool image on the printout, altering the title, including a schematic in the report, selecting values for display in the tool summary, and adjusting the size of images.

*Click the **Refresh** button prior to printing to see any changes reflected in the Print Preview screen.*

By default, the title for the report is the name of the tool; however this can be modified by entering an alternate title in the **Title** box.

Click the **Draw to Scale** togglebox to produce a printout in which the images associated with each tool in the string are stretched relatively (in size) to one another.

To adjust the scale of the image displayed on the printout, enter a value from .5 (half size) to 5 in the **Image scale width** or **Image scale height** box.



Chapter 13
Using the Well Editor

Well Editor allows the configuration of a well, including casing, tubing, and liner geometry. Well Editor stores all of the wells in a single database and is accessible across the software suite, providing a consistent interface between applications.

When configured, wells are used for many calculations. Note that a well is not required for basic fatigue calculations. The following links point to paragraphs that cover these topics in greater detail.

- "Starting Well Editor" on page 302.
- "The Well Editor Display Screen" on page 303.
- "Working with Wells" on page 307.
- "Configuring Wells" on page 312.
- "Troubleshooting in Well Editor" on page 326.

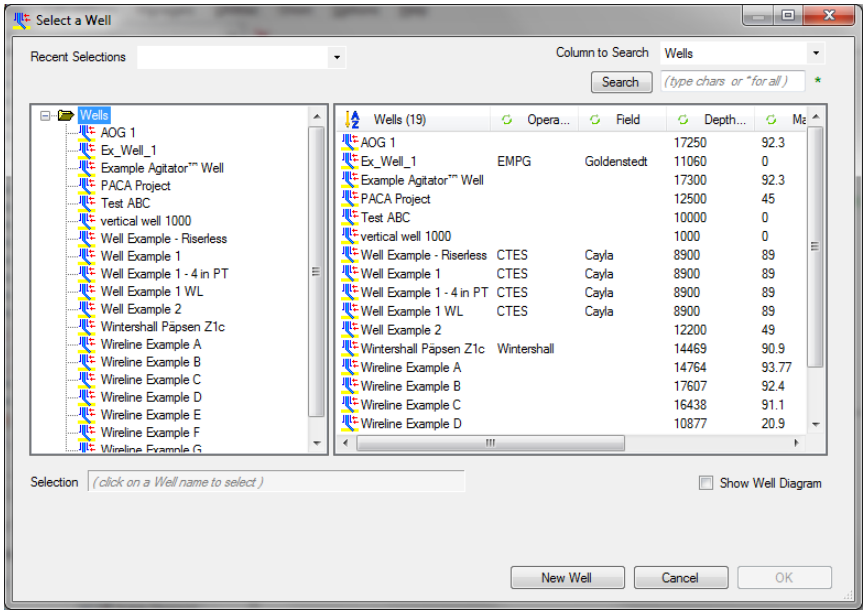
Starting Well Editor

The following procedure outlines how to start Well Editor.

➢ To start Well Editor

1. Find the Cerberus Main Menu (see page 21).
2. Select, from the menu bar, **Managers > Well Editor**.
3. The Select a Well dialog appears, from which the user chooses a well.

Select a Well



OR

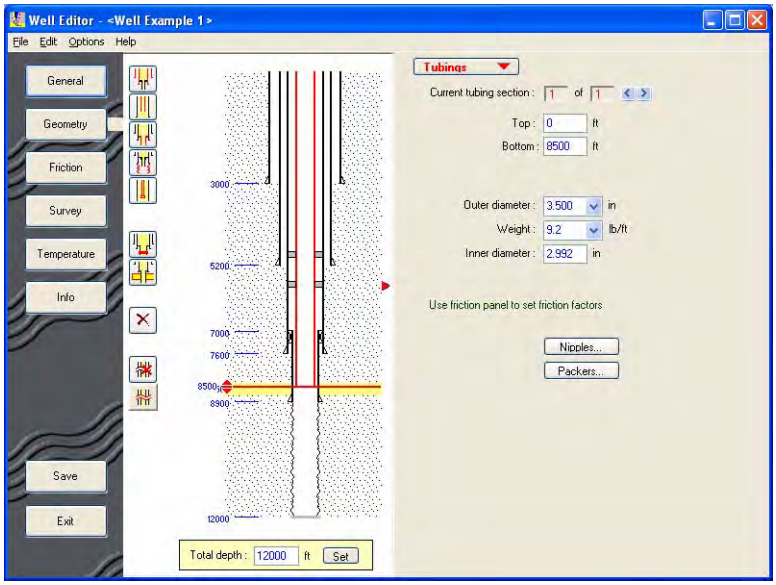
Launch Well Editor from within other applications in the Cerberus suite by clicking the **Well Editor** button .

The Well Editor Display Screen

The Well Editor display screen contains the following areas:

- **Menu -** Quick access to common tasks and setting preferences.
- **Well Diagram** - A diagram of the well. Click on a tubular in the diagram to view or edit parameters.
- **Input Panels -** Click the red dropdown list to view or edit the well geometry.

Well Editor Main Display Screen



The Well Editor Tools

The Well Editor has a complete set of tools to simplify the modeling of a well. The following sections cover these tools in greater detail.

The Main Well Editor Toolbar

Below is a description of the different tools available on the main Well Editor display screen.

Well Editor Tools

Tab	Button	Function
General		Enter total depth, well location, reference point, and height. Comments can be entered as well. Also used to view TVD.
Geometry		Add and configure casing and associated nipples (see page 317).
		Add and configure tubing and associated nipples and packers (see page 316).
		Add and configure liners and associated nipples (see page 317).
		Add and configure open hole (see page 317).
		Add and configure drill pipe sections
		Add and configure a bridge plug (see page 317).
		Add and configure a reservoir (see page 317).
		Delete the current selection.
		Clear all breaks in the well.
		Add a break - exclude a section of the well view.

Well Editor Tools

Tab	Button	Function
	Friction	Opens the friction-factor data grid.
	Survey	Opens the survey data for the current well (see page 318).
	Temperature	Opens the temperature profile for the current well (see page 324).
	Info	Opens the well information display screen.
	Save	Saves the current well to disk.
	Exit	Exits Well Editor.



Rest the cursor over a tool button for a few seconds for the tool name to appear.

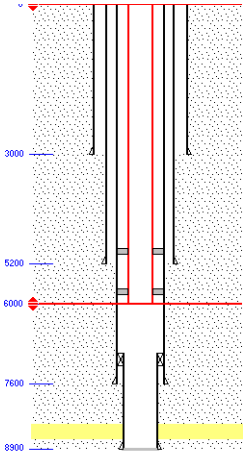
Viewing a Well

Well Editor displays information about the well on the Well Diagram in the center of the main screen. The diagram shows all tubing sections, liners, casing, open hole, and reservoir, as well as the **effective well**, which is used for calculations. The well always appears vertically in this diagram, even if it is a deviated well. .

To . . .	Do this . . .
See inputs for an area of the well.	Click specific area (such as a liner) or simply click the appropriate tab on the left. The inputs appear on the right side of the screen.
Zoom in and out of the well geometry diagram.	Click and drag a rectangle around the area to zoom in on. Right-click the well to zoom out.

To . . .	Do this . . .
View "smart" labeling on the well plot.	Click the red arrow to the right of the diagram.
View the effective well.	Choose **Show Effective Well** from the **Options** menu.
Set friction independent of geometry.	Click the **Geometry** tab, then the **Advanced...** button on the input screen.
Set friction defaults.	Choose **Friction Defaults...** from the **Options** menu.
Change units.	Choose **Units...** from the **Options** menu.

Well Diagram



Viewing Survey Data Graphs

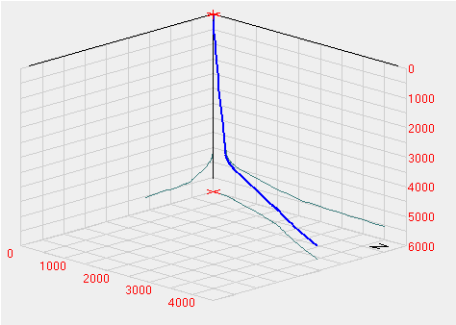
The Survey Graph is a three dimensional representation of the survey path and is viewable from the Survey tab. The graph automatically updates to show the most recent survey data. The graph can show the following items:

- Well path
- Axes (North, South, East, West, and depth)
- Depth indicator (for current row in survey data)
- Shadows (well path projected onto plane)

- Grid lines (of plane)

To . . .	Do this . . .
Turn auto-rotating on and off.	Right-click the diagram. Select **Auto-Rotate** from the list.
Adjust the elevation and rotation angle.	Turn off auto-rotate, then click the arrows below the graph.
View the graph from a standard elevation and rotation angle.	Turn off auto-rotate. Then right-click the diagram and select isometric view from the list.
Zoom in on the graph.	Draw a rectangle around the area to zoom in on.
Zoom back out of the graph.	Right-click the graph, then select **Zoom Out**.
Customize the appearance of the graph.	Right-click the graph. Select **Options** from the menu. Select the options. Then click **Graph** to return to the graph.
View the survey plot only.	Click the red arrow to the right of the graph or deactivate the **View Data** button. Click again to show the data screen.

Survey Graph



Working with Wells

Creating a New Well

Each well must have a unique name. Well Editor stores all of the wells in one database.

Create a new well manually or use Quick Well (see page 313) to quickly preselect certain components in the well.

➢ To create a new well manually

1. Start Well Editor.
2. Select **File** > **New**.
3. Enter the well name.
4. Click **OK**.
5. Click the **Skip** button.
6. Configure the **well** (see page 363).

Opening an Existing Well

➢ To open an existing well

1. Select **File** > **Open** from the menu bar.
2. Click the name of the well.
3. Click **OK**.

Saving a Well

Saving a well to the hard drive records changes to the well in the Wells database. There are two ways to save a well:

- Select **File** > **Save** from the menu bar to save a well with the current name.

 OR

- Select **File** > **Save as...** from the menu bar to save a well with a new name.

Deleting a Well

Unnecessary wells can be deleted. However, when deleted, the information cannot be recovered.



If the deleted well is currently open, Cerberus prompts the user to select a new well to open.

➢ To delete a well

1. Select **File** > **Delete...** from the menu bar.
2. Select one or more wells from the list.
3. Click **OK**.
4. Click **Yes** to confirm.

Importing a Well

The user may import or export well information from other computers. Export the well on the original computer to removable media. Then, place the removable media in the other computer and import the well.

To . . .	Do this . . .
Select a group of items that are next to each other.	Click the first item, then hold down the SHIFT key and click the last item.
Select multiple non-adjacent items.	Hold down the CTRL key and click each item.

➢ To import a well

1. If importing from removable media, insert the media into the drive.
2. Start Well Editor.
3. On the menu bar, select **File** > **Import**....
4. Locate the drive and path where the well is located.
5. Select the names of the wells to import.
6. Click the **Open** button.
7. Click **OK**.

Exporting a Well

Well information from other computers can be exported and imported. Export the well on the original computer to removable media. Then, put the removable media in the other computer and import the well.



To copy only the survey data, import and export the data as a text file from the Survey screen in Well Editor.

To . . .	Do this . . .
Select a group of adjacent items	Click the first item; then hold down the SHIFT key and click the last item.

To . . .	Do this . . .
Select multiple non-adjacent items	Hold down the CTRL key and click each item.

➢ To export a well

1. If exporting to removable media, insert the media into the drive.
2. Start Well Editor.
3. From the menu bar, select **File** > **Export...**.
4. Select the name(s) of the well(s) to export.
5. Locate the drive and path to which to export the well or wells.
6. Click **Save**.
7. Click **OK**.

Printing Well Data

Print detailed reports, including the well's configuration, from Well Editor. Print data to a printer or to a PDF file for emailing or viewing by others.

The free Adobe Reader must be installed to view PDF files. Download the reader from Adobe's website at www.adobe.com

NOV CTES does not recommend using any PDF printer other than the NOV CTES PDF generator. Doing so may result in unpredictable and undesirable results.

Usually the survey graph prints in isometric view. To print the plan view or side view, print a custom view (see page 362).

➢ To print well data to a printer

1. Select **File** > **Print** from the menu bar.
2. Select **Include Cover Page** to include a cover page in the report.
 a. Click the **Configure** button to specify the cover-page elements. Click **Done** when the desired information is entered into the related fields.
3. Select the data to include in the report.

 In the Print Data section, use any or all of several options for data:

- **Well Diagram** - Includes a diagram of the well.
- **Well Data** - Includes data on the well.
- **Survey Data** - Includes the survey data for the well.
- **Survey Graphic (Isometric View)** - Includes an isometric graphic of the well. Additionally, **current elevation and rotation** can also be included in the graph.

4. Select the printer options.

 Select the printer:

 - Click the ellipses button [...] associated with the **Printer** field.

 A standard Windows-operating-system Print dialog appears.
 - Select the desired print specifications and click **OK**.

 *The user may select **CTES PDF Printer** from the drop-down menu to print a PDF file.*

 A logo (if not previously configured) can be selected:

 - Click the ellipses button [...] associated with the **Logo** field.

 A standard Windows-operating-system Open dialog appears.
 - Select the desired logo.
 - Click **Open**.

5. Click **Print**.

Printing a Custom View of the Survey Path

The survey graph is depicted in a standard view when the Survey tab is selected in Well Editor. Change this view and print, as desired, using the keys at the bottom of the screen.

➢ To print a custom view of the survey graph

1. Click the **Survey** tab.

 The Well Editor screen is repopulated to depict a diagram of the open well.

2. Click **Rotate** to turn off auto-rotate.

3. Adjust the **position** of the well graph.

 Use the three arrow keys to adjust the appearance of the well diagram.

 - Use the **Position arrows**  to position the well diagram on the page.
 - Use the **Zoom arrows** to zoom in or out on the well diagram.

- Use the **Rotate arrows**  to rotate the well diagram along four axes. This control allows for any side view of the well, as well as bottom-up or top-down view.

4. Select **File** > **Print...** from the menu bar.

 The Print Well dialog appears.

5. Click the **Survey Graph** checkbox.

6. Select **Use current elevation and rotation**.

7. Click **Print**.

 The graph is sent to the selected printer.

Emailing Well Data Files

Export and email well data files to other Cerberus users after the files are exported (see page 374) or printed as a PDF (.pdf) file (see page 361).

Configuring Wells

Configure Wells manually in Well Editor or use Quick Well to pre-select items (see page 364).

Regardless of how the well is configured, NOV CTES recommends the following configuration sequence:

1. "Configuring Depth" on page 313.
2. "Configuring Casing" on page 315.
3. "Configuring Liners" on page 315.
4. "Configuring Tubing" on page 316.
5. "Configuring a Drill Pipe" on page 316.
6. "Configuring the Open Hole" on page 317.
7. "Configuring the Bridge Plug" on page 317.
8. "Configuring the Reservoir" on page 317.
9. "Configuring Friction" on page 318.
10. "Configuring Temperature" on page 324.

NOV CTES also recommends entering the following well data for record keeping and reporting purposes, although it is not used in calculations:

- "Configure Location and Miscellaneous Information" on page 325.

Using Quick Well

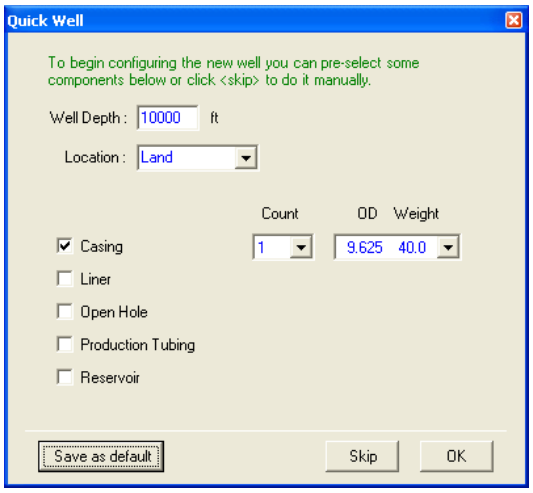
Quick Well can be used to create and configuring a new well by preselecting components of the well. Quick Well automatically opens when a new well is created. Common configurations can be saved as the default.

➢ To configure a new well using Quick Well

1. Start Well Editor.
2. Select **File > New...** from the menu bar.
3. Enter the name of the new well.
4. Click **OK**. The Quick Well dialog appears.
5. Enter the well configurations.

 To save the input as the default well, click the **Save as Default** button; then click **OK**.

Quick Well



Configuring Depth

The well depth is set within the Well Editor General Tab of the main Well Editor display screen. Setting the location, reference point, and height are also important. To decrease the effective depth of the well, specify a bridge plug.

Enter the total well depth before entering other properties. Well Editor uses the well depth as part of its error-checking process when other properties are entered. Avoid changing the well depth after other properties are entered.

➢ To configure total well depth

1. In Well Editor, select the **General** tab.
2. Enter the **Total depth (MD)** of the well.

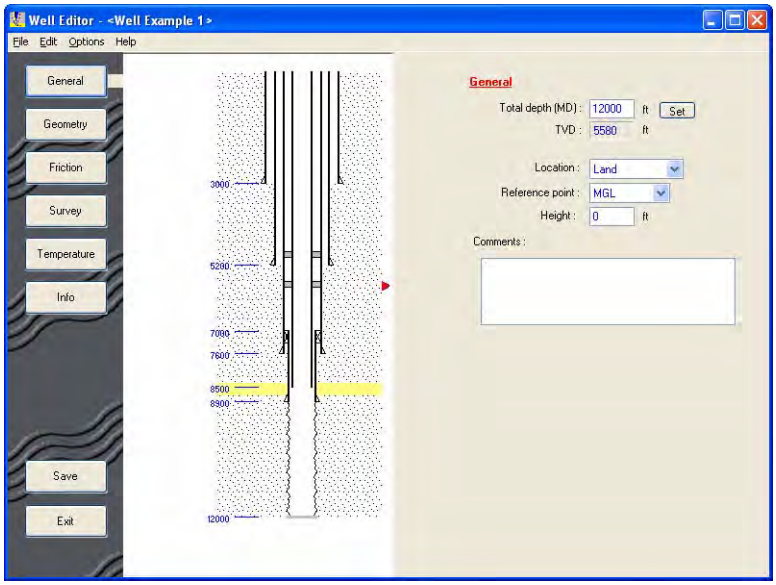
3. Click **Set**.

4. Enter the other well data.

 This data includes **location**, depth **reference point** information for the well, the **height**, and any **Comments**.

 Height refers to the distance between the reference point and the zero (0) depth of the well.

 Well Editor General Tab



Configuring Subsea Information

Subsea wells present specific configuration requirements. Wireline subsea wells may be riserless.

➢ To configure subsea wells

1. In Well Editor, select the **General** tab.

 *If the well is new, launch the Quick Well utility (see "Using Quick Well" on page 313); select **Subsea** as the **Location**.*

2. For the **Location** field, select **Subsea** from the dropdown.

 a. If the project is a Wireline project, the well may be riserless. If this is the case, click (to check) **Riserless** (this field is located adjacent to the **Location** field).

 *The **Riserless** option should be available for selection for a Wireline project in Quick Well.*

3. Input or edit the **Riser geometry**.

 a. If the well is **Riserless**, use the default values for new wells (already present), or input the **Ocean density**, **Ocean viscosity**, and **Ocean current data**.

4. Save the Well information.

5. Complete the Run at Depth calculation (see "Run at Depth" on page 94 for additional information).

Configuring Casing

Casing sections of the well, as well as associated nipples, are defined within the Geometry Tab – Reservoir Input. A new well starts out with one casing string. Well Editor displays this information on the well diagram.

To . . .	Do this . . .
Find the internal diameter.	Select the **Nominal Diameter** and **Casing Weight**. Well Editor looks up the internal diameter.
Enter nipples for the current casing section.	Click **Nipples...**.
Set friction independent of geometry.	Click **Advanced...**.

➢ To configure casing

1. Select the **Geometry** tab in Well Editor.
2. To add casing, click **Add Casing** .
3. Enter the casing properties.

 To add additional casing, repeat Steps 2 and 3.

Configuring Liners

Liners in the well, as well as associated nipples, are defined within the Geometry Tab – Reservoir Input. Well Editor displays this information on the well diagram.

To . . .	Do this . . .
Find the internal diameter.	Select the **Nominal Diameter** and **Liner Weight**. Well Editor looks up the internal diameter.
Enter nipples for the current liner section.	Click **Nipples**.
Set friction independent of geometry.	Click **Advanced**.

➢ To configure liners

1. Select the **Geometry** tab in Well Editor.
2. To add a liner, click **Add Liner** .
3. Enter the liner properties.

 To enter additional liners, repeat Steps 2 and 3.

Configuring Tubing

Production tubing of the well, as well as associated nipples and packers, is defined within the Geometry Tab – Reservoir Input. Well Editor displays this information on the well diagram.

To . . .	Do this . . .
Find the internal diameter.	Select the **Nominal Diameter** and **Tubing Weight**. Well Editor looks up the internal diameter.
Enter packers for the current tubing section.	Click **Packers...**.
Enter nipples for the current tubing section.	Click **Nipples...**.
Set friction independent of geometry.	Click **Advanced...**.

➢ To configure tubing

1. Select the **Geometry** tab in Well Editor.
2. To add tubing, click the **Add Tubing** button .
3. Enter the **tubing properties**.

 Add additional sections as appropriate, if the physical geometry varies.

Configuring a Drill Pipe

Though drill pipe is not normally considered part of a well, Cerberus provides the ability to add drill pipe to the well to allow modeling of a job performed inside drill pipe.

➢ To configure a drill pipe

1. Select the **Geometry** tab in Well Editor.
2. Click **Add Drill Pipe** .
3. Enter the drill pipe properties.

Configuring the Open Hole

If the depth of the deepest tubular (tubing, casing, or liner) is less than the total well depth, the remaining area must be defined on the Geometry Tab – Reservoir Input. The first open hole section should start at the bottom of the deepest tubular. The last open hole section should extend to the total well depth. Well Editor displays this information on the well diagram.

If multiple open hole regions are entered, their diameters must decrease with depth.

➢ To configure the open hole

1. Select the **Geometry** tab in Well Editor.
2. To add an open hole section, click **Add Open Hole** .
3. Enter the open hole properties.

Configuring the Bridge Plug

If the well has a bridge plug, it can be specified within the Geometry Tab – Reservoir Input. Well Editor displays the location of the bridge plug on the well diagram. Adding a bridge plug decreases the effective depth of the well.

➢ To configure the bridge plug

1. Select the **Geometry** tab in Well Editor.
2. Click **Add Bridge Plug** .
3. Enter the **Depth** of the bridge plug.

 The well graph reconfigures to show the location and presence of the bridge plug.

Configuring the Reservoir

The reservoir location, pressure, and reservoir model inputs are defined on the Geometry Tab – Reservoir Input. Well Editor displays the location of the reservoir on the well diagram.

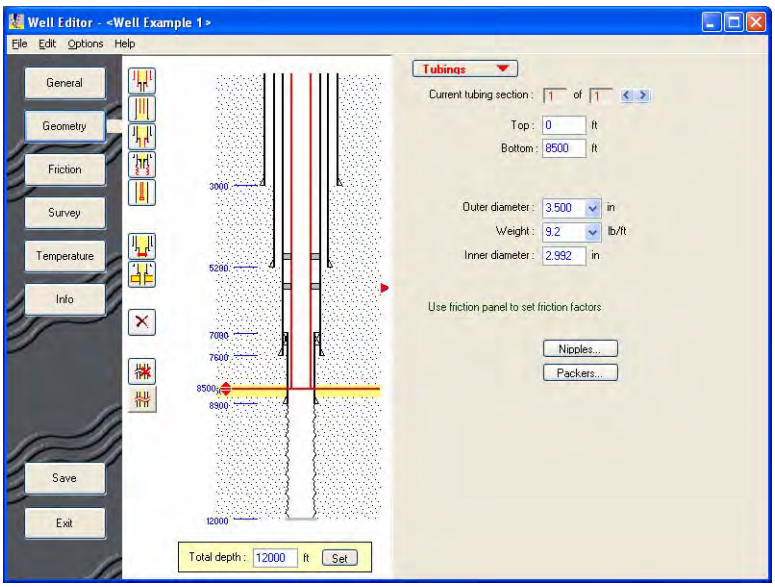
The user may experiment with different reservoir inputs directly in Hydra. However, only the inputs entered here are saved with the well.

To . . .	Do this . . .
Use a fluid that does not appear in the list.	Click the Fluid Manager button. Configure and save the fluid in Fluid Manager.
Have Hydra calculate bubble point pressure.	Using the Darcy Oil reservoir model, leave the bubble point pressure zero.

➢ To configure the reservoir

1. Select the **Geometry** tab in Well Editor.
2. Select **Reservoir** from the main dropdown button.
3. Enter the location, fluid type, and pressure of the reservoir.
4. Click the **Advanced** button to select the well's **reservoir model** from the drop-down menu.
5. Enter the properties for the selected reservoir model.

Geometry Tab – Reservoir Input



Configuring Friction

There are two ways to set friction factors, one of which is to set friction based on effective well geometry; the other is to set friction independently of geometry.

➢ To configure friction

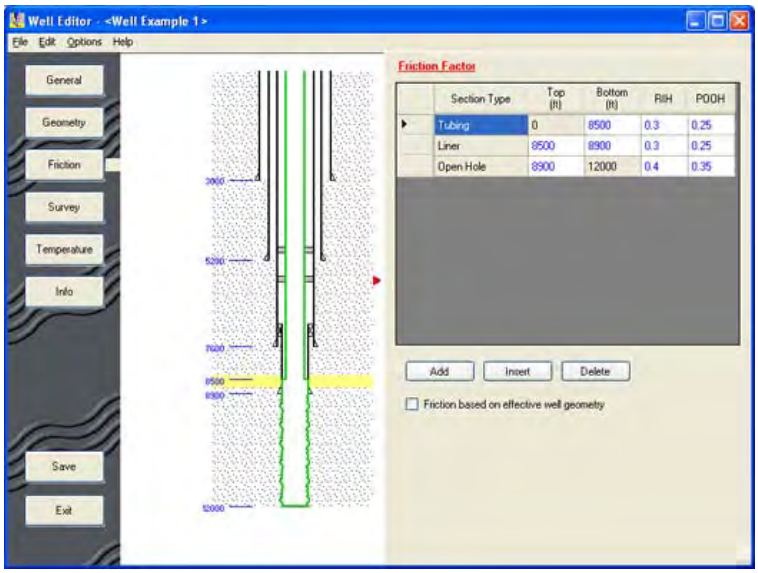
1. In Well Editor, select the **Friction** tab.

2. Click (to check or to uncheck) **Friction based on effective well geometry**.

3. If friction is *not* based on effective well geometry, the user may **Add**, **Insert**, or **Delete** friction-factor information. If friction *is* based on effective well geometry, friction-factor information cannot be added because start- and end-depth information is predetermined:

 - Click **Add** to add a new row to the end of the datagrid.
 - Click **Insert** to insert a new row before the selected row.
 - Click **Delete** to delete the selected row.
 - Left-click within an RIH or POOH cell; then right-click to set the friction factor to a single value in all the rows below the selected cell.

Well Editor Friction Tab



Within Orpheus, friction-factor information is found in the Downhole Conditions portion of the Calculate tab.

Configuring Survey Data

The survey path is defined by entering survey points (depth, inclination, and azimuth) on the Survey Data Display Screen. Well Editor joins the points with a curve (not a straight line) and displays this information in a three-dimensional graph. Well Editor also calculates the dogleg severity and true vertical depth. There are two ways to open the survey data:

- Click the **Survey** tab in Well Editor.

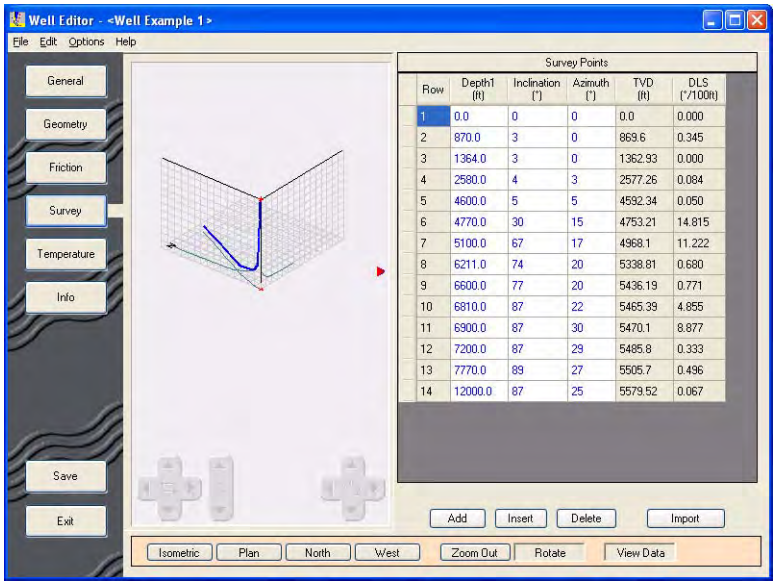
 OR

- Select **Survey** from the **Edit** menu.

Input survey data manually, paste the data from a spreadsheet, or import data from a text file. Graphs are viewable after entering or importing the survey data (see page 355).

To . . .	Do this . . .
Add a row above the current row.	Click **Insert**.
Add a row after the last row.	Press ENTER at the end of the last row.
Delete a row.	Select the row or rows to be deleted. Click **Delete**.

Survey Data Display Screen



Entering Survey Points

To enter data in a row, simply type within the table.

Using the Survey Wizard

Use the Survey Wizard to quickly and easily generate common well-path configurations.

➢ To use the Survey Wizard

1. Select the **Survey** tab on the Well Editor panel.
2. Click the **New Survey...** button on the data screen; then click the **Survey Wizard** button.
3. Select the well type.

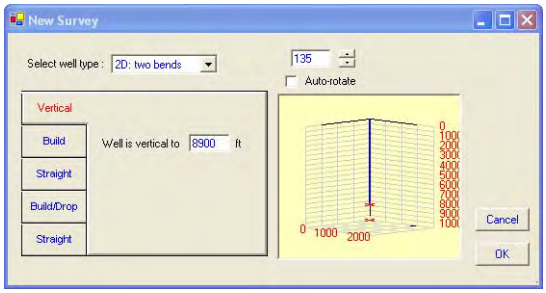
4. Enter the required data.

 Depending on the selection, the depth, inclination, and azimuth may need to be entered. The 3-D graph changes to reflect the new configuration.

5. If the results are satisfactory, click **OK**.

 A warning dialog appears indicating the existing survey data will be overwritten. Click **Yes** to proceed or **No** to cancel.

Survey Wizard



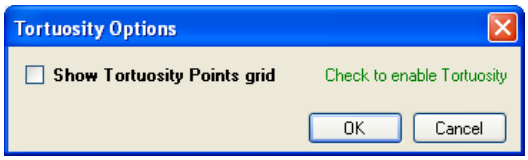
Adjusting Survey Tortuosity

Tortuosity is the “spiraling” or “corkscrew” effect that occurs as wells are drilled. Both “macro” and “micro” tortuosity occur. Macro tortuosity represents the difference in dogleg severity/tortuosity between the planned well survey and the final well survey. Micro tortuosity (or spiral tortuosity) occurs on a much smaller scale than macro tortuosity, so micro tortuosity does not appear in conventional survey data. Cerberus provides the means to adjust micro tortuosity/increase dogleg severity, and allows its appearance in the survey data.

➢ To adjust micro tortuosity

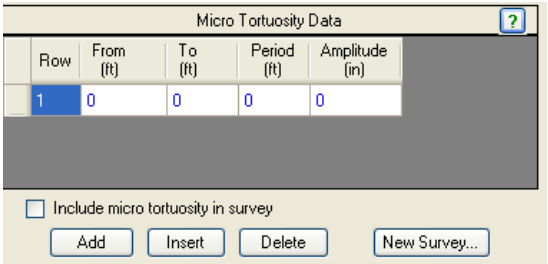
1. Within the Well Editor, select **Options** > **Survey Tortuosity...** from the menu bar.

 The Tortuosity Options dialog appears.



2. Click (to check) the **Show Tortuosity Points grid** togglebox.

 The **Micro Tortuosity Data** grid appears within the **Survey** tab, underneath the **Survey Points** grid. Initially, only one tortuosity point appears (each point corresponds to one row).



3. Input the depth interval (**From – To**), and the **Period** and **Amplitude** parameters corresponding to each numbered row.

In reality, the **Period** *of the spiral ranges between two and 10 feet, while a typical* **Amplitude** *is 0.5 inch.*

 a. **Add**, **Insert**, or **Delete** points/rows using the corresponding buttons.

4. Adjust the **Period** and **Amplitude** to match each dogleg of interest.

5. Click the **Save** tab.

The micro tortuosity data is only saved in relation to the current project session and is not saved with the well. If modeling with tortuosity is desired, the user must re-enter information into the Well Editor each time the project is opened.

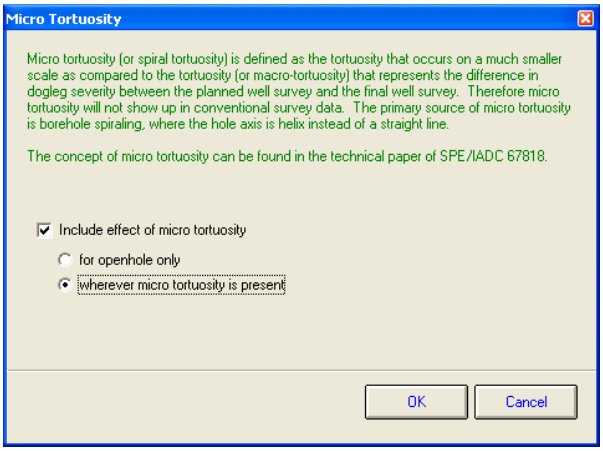
At this juncture, the user must "tell" Orpheus to utilize the additional survey points when performing calculations.

6. Start Orpheus.

7. Select the Options tab; then Features and Effects.

8. Click to toggle **Include Additional Micro Tortuosity**; then click the associated ellipses button.

 The Micro Tortuosity dialog appears.



9. Ensure that the **Include effect of micro tortuosity** box is checked.

10. Click one of the toggleboxes to indicate whether the effect of micro tortuosity will be considered **for openhole only**, or **wherever micro tortuosity is present**.

11. Click **OK**.

Exporting Survey Data

Share survey data with other people by exporting the data to a text file format. The survey data can be printed separately from the Well data or included as part of the Well data report.

*Survey data is normally included with any well export function from all tabs within Well Editor. To export ONLY survey data, the Survey tab MUST be selected and the user MUST select **Export Survey...**. Selecting **Export...** will include all other well data.*

➢ To export survey data only

1. Select the **Survey** tab.
2. Select **File** > **Export Survey...** from the menu bar.
3. Enter the **file name** and **path** to which to export the data.
4. Click **Save** to export the data.

➢ To export survey data with well data

1. Select **File** > **Export...** from the menu bar.
2. Select the desired well to export.
3. Click **OK**.
4. Select the **path** and **file name** for the well data.

 If necessary, navigate to the desired location.

 If desired, enter a new file name for the file.

 Ensure the file type is Cerberus File (*.zcx). Otherwise, Cerberus will not be able to read the file and import it.
5. Click **Save**.
6. Click **OK**.

Importing Survey Data

Share survey data with other people by importing the data to a text file format.

➢ To import survey data

1. Select the **Survey** tab on the panel.
2. Click the **New Survey...** button on the data screen.

 The New Survey dialog appears.
3. Click **Import from File** 

.

 The New Survey dialog box is repopulated to show the format accepted by Cerberus.

4. Click **Import from File** 

.

 A warning dialog box appears stating the depths will be imported according to the Units setting. In other words, if units are set to Metric, then the depth is imported as Metric.

5. Click **Yes**.

 A warning dialog box appears, stating the current survey data will be cleared.

6. Click **Yes**.

 The Import Survey Data dialog appears.

7. Select the file name and path from which to import the data.

8. Click **Open**.

 The Import Survey Data Import is complete dialog appears.

9. Click **OK**.

 The data is imported, with the depth imported in the selected Units setting and the previous survey data cleared. The user is returned to the Well Editor main screen.

Configuring Temperature

The surface temperature and temperature gradient are defined within the Temperature tab. Well Editor calculates the downhole temperature.



The user may also experiment with different temperature profiles directly in Hydra. However, only the inputs entered under the Temperature tab in Well Editor are saved with the well.

To . . .	Do this . . .
Enter the temperature in different units (such as Fahrenheit to Celsius).	Click **Options** and select **Units...** from the menu. The units conversion utility can be used to calculate the value in another unit system.
Check the downhole temperature.	Enter a measured depth. Well Editor converts the measured depth to true vertical depth (based on the current survey data), then applies the temperature gradient.

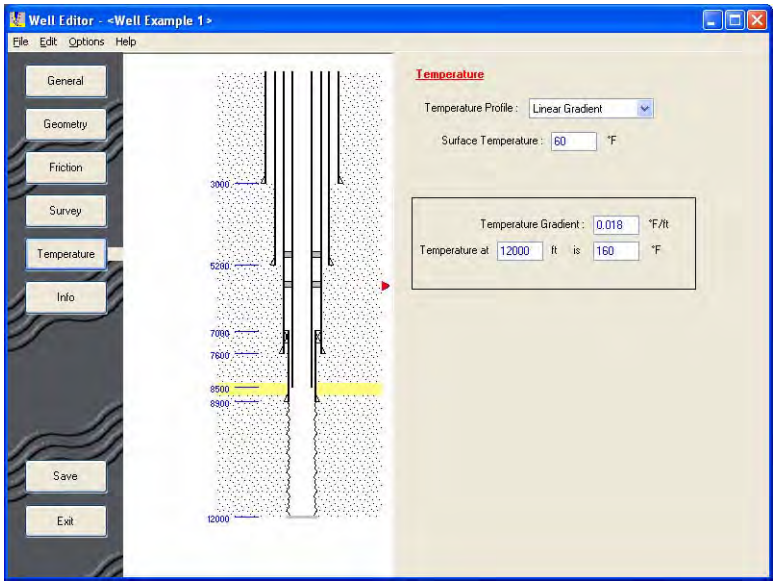
➢ To configure temperature properties

1. Select the **Temperature** tab in Well Editor.

2. Select the **Temperature Profile** from the drop-down menu.

3. Enter the **Surface Temperature** and **Temperature Gradient**.

Temperature tab



Configure Location and Miscellaneous Information

The location of the well, as well as its owner, elevation, latitude, and longitude, are configurable on the Info tab. This information is used on reports and kept with the records. The properties do not affect calculations. However, the H2S and CO2 inputs are used in corrosion calculations. If corrosion is being tracked, configure these inputs.

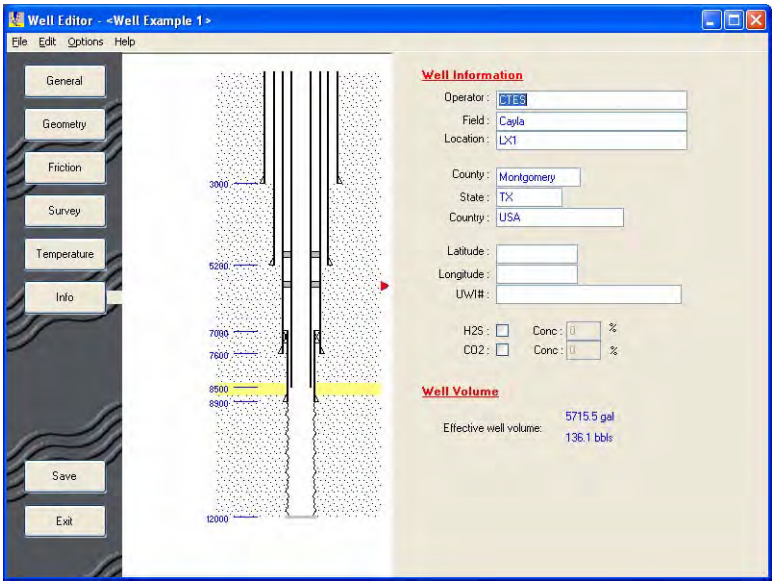
When a well is selected in Job Manager, this information is copied into the well physical data for the job.

➢ To configure location

1. Select the **Info** tab in Well Editor.

2. Enter the information.

Info tab



Troubleshooting in Well Editor

Situation	Possible Cause	Action
I don't see the reservoir fluid I want in the list.	The reservoir fluid was deleted from or not included in the list of fluids configured in Fluid Manager.	Configure the fluid in Fluid Manager, then pick the fluid for the well.
	Fluids defined as a foam are not included in the list.	Redefine the fluid type in Fluid Manager.

Chapter 14
Using Reel Editor

Reel Editor is used to configure the information about the reel used to hold the coiled tubing. Reel Editor stores all its reels in a single database. Because a reel will hold many different strings during its working life, it is treated separately from the CT string. The following links point to paragraphs that cover these topics in more detail.

- "Starting Reel Editor" on page 327.
- "The Reel Editor Screen" on page 328.
- "Working with Reels" on page 329.
- "Configuring Reels" on page 335.
- "Troubleshooting in Reel Editor" on page 338.
- "Troubleshooting in Reel Editor" on page 338.

Starting Reel Editor

➢ To start Reel Editor

1. Find the Cerberus Main Menu.
2. Click one of the following:
 - The Managers menu-bar item
 - The Hydra icon
3. If the Hydra icon was clicked, select the **Reel Editor** toolbar button .

 Reel Editor will now start.



Reel Editor can be launched from other applications in the Cerberus suite by clicking the Reel Editor button .

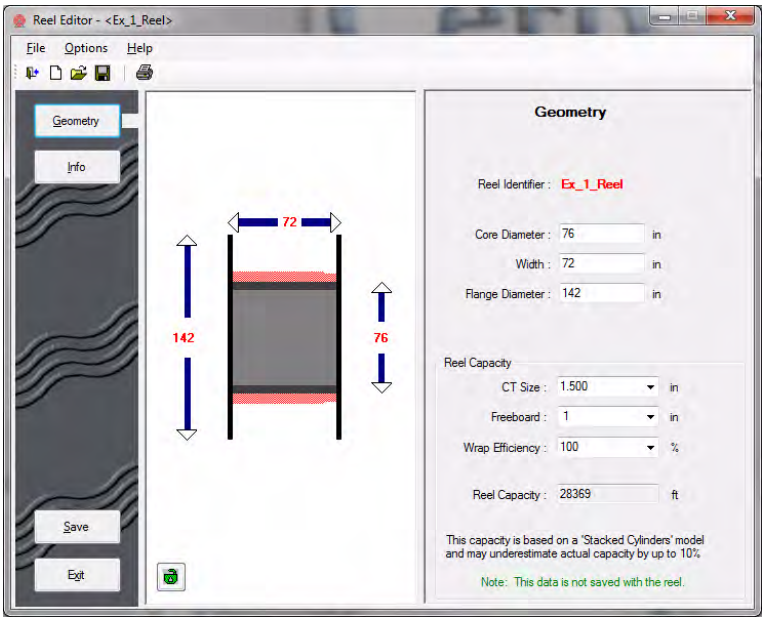
The Reel Editor Screen

The Reel Editor Main Screen -- Geometry Tab contains the following areas:

- **Toolbar -** Click these buttons for quick access to common tasks. See “The Reel Editor Toolbars” on page 328.
- **Geometry Tab -** Contains two panels: one showing reel dimensions and lock status, and one showing reel geometry. In the Geometry panel, enter the reel geometry (core diameter, width, and flange diameter) and the capacity of the reel.
- **Info Tab -** Similarly to the Geometry tab, the Info tab contains two panels. Enter Service Data and the configuration of the reel (work reel or shipping spool).

To...	Do this...
Show or hide archived reels. (See “Archiving a Reel” on page 334.)	From the Select a Reel dialog, click (to check or uncheck) **Show Archived Reels**.
Show or hide work reels. (See “Configuring Reel Type” on page 336.)	From the Select a Reel dialog, click (to check or uncheck) **Show Work Reels**.
Show or hide shipping spools. (See “Configuring Reel Type” on page 336.)	From the Select a Reel dialog, click (to check or uncheck) **Show Shipping Reels**.

Reel Editor Main Screen -- Geometry Tab



The Reel Editor Toolbars

Below is a description of the buttons on the main Reel Editor toolbar, and each button’s function.

Rest the cursor over a tool button for a few seconds for the tool name to appear.

Reel Editor Main Toolbar

Tool Button	Tool Name	Function
	Exit	Exits Reel Editor.
	New	Opens the New Reel dialog box to create a new reel. See “Creating a New Reel” on page 330.
	Open	Opens the Select Reel dialog box to open an existing reel (see page 330).
	Save	Saves the current reel to disk.
	Print	Opens the Report Editor dialog box to print the current reel. See “Printing Reel Data” on page 334.

Working with Reels

The following links point to paragraphs that cover these topics in greater detail.

- “Creating a New Reel” on page 330.
- “Opening an Existing Reel” on page 330.
- “Saving a Reel” on page 330.
- “Deleting a Reel” on page 331.
- “Importing a Reel” on page 332.
- “Exporting a Reel” on page 332.
- “Locking a Reel” on page 333.
- “Unlocking a Reel” on page 333.
- “Archiving a Reel” on page 334.
- “Printing Reel Data” on page 334.

Creating a New Reel

Each reel must have a unique name. Because Reel Editor stores all of the reels in one database, there is no need to enter a database name.

➢ To create a new reel

1. Start Reel Editor.
2. Click **Create NewReel**  on the toolbar.

 The New Reel dialog appears.
3. Enter the new-reel name. The user may also click the **Show Existing** button to determine the reels already in use.
4. Click **OK**.

 Reel Editor starts the new reel with default values.
5. Configure the reel.

 See "Configuring Reels" on page 335.

Opening an Existing Reel

After a reel is created, it can be opened and used.

➢ To open an existing reel

1. Click **Open Existing Reel**  on the toolbar.

 OR

 Select **File > Open Reel** from the menu bar.

 The Select a Reel dialog appears.
2. Select the reel to open, and click **OK**.



If a reel isn't listed, try showing archived reels, work reels, and shipping spools.

3. The Select a Reel dialog box closes and the user is returned to the main Reel Editor screen (Geometry tab), which is populated with the new-reel information.

Saving a Reel

Reels are saved in order to record changes to the reel on the hard drive. A reel can be saved without renaming (Save) or saved with a new name (Save As).

➢ To save a reel

- Click **Save**  on the toolbar.

OR

1. Select **File** > **Save Reel As...** from the menu bar.

 The Save as... dialog appears.

2. Enter a **name** for the reel.

 Names can be up to 32 characters long, with letters, numbers, and spaces.

3. Click **OK**.

 The **Save as...** dialog is closed, the reel is saved under the new name, and the user is returned to the main Reel Editor screen.

Deleting a Reel

Unnecessary reels can be deleted.

 *When a reel is deleted, the information cannot be recovered. The reel can be archived instead. See "Archiving a Reel" on page 334.*

➢ To delete a reel

1. Select **File > Delete Reel** from the menu bar.

 The Delete Reels dialog appears.

2. Select the desired reel.

 If the reel is locked, unlock it. See "Unlocking a Reel" on page 333.

3. Click **OK**.

 A Delete Reel warning box appears, stating the operation can not be undone to confirm deleting the reel.

4. Click **Yes** to confirm deleting the reel.

 The reel is deleted and the user is returned to the main Reel Editor screen.

Importing a Reel

Reels can be shared between computers by importing and exporting them. Reels are exported to removable media for transferring to another computer. Once on removable media, a reel can be imported into Reel Editor.

To...	Do this...
Select a group of items that are next to each other.	Click the first item, then hold down the SHIFT key and click the last item.
Select multiple non-adjacent items.	Hold down the CTRL key and click each item.

➢ To import a reel

1. If a reel is being imported from removable media, insert the removable media in the computer.
2. Start Reel Editor.
3. Select **File** > **Import...** from the menu bar.

 A Windows interface appears, with a top-bar instruction to **Select a reel project to import**.
4. Select the drive and folder for the Cerberus File (ZCX or ZCERB), Access Database (MDB), Cerberus Download File (CDF), or Zip file (ZIP).
5. Select the reel(s) to import.
6. Click **Open** or **Open as read only**.

 Reel Editor imports the information.

Exporting a Reel

Reels can be shared between computers by importing and exporting them. A reel is exported to removable media for transferring to another computer. If a reel is on removable media, the reel can be imported into Reel Editor.

To...	Do this...
Select a group of items that are next to each other.	Click the first item, then hold down the SHIFT key and click the last item.

To...	Do this...
Select multiple non-adjacent items.	Hold down the CTRL key and click each item.

➢ To export a reel

1. Start Reel Editor.
2. Select **File** > **Export...** from the menu bar.

 The Export Reel dialog appears.
3. Select the reel(s) to export
4. Click **OK**.

 The **Save As** dialog appears.
5. Select the destination drive and folder.
6. Click **Save**.

 Reel Editor exports the reels database to the specified path.

 When finished, the Reel Editor Export is complete dialog box appears.
7. Click **OK**.

Locking a Reel

Reels can be locked to prevent inadvertently modifying or deleting them.

To quickly see if a reel is locked, look for the Lock in the lower left corner of the screen.

➢ To lock a reel

1. Click the **Lock** icon in the lower left-hand portion of the Geometry or Info tab. Clicking the icon acts as a toggle to lock or unlock the reel, depending on the state of the reel when clicked upon.

 OR
2. Select **Options > Locked** from the menu bar (if the menu-bar item is unshaded and unchecked).

Unlocking a Reel

A reel must be unlocked before any changes can be made or before it can be deleted.

➢ To unlock a reel

1. Click the **Lock** icon in the lower left-hand portion of the Geometry or Info tab. Clicking the icon acts as a toggle to lock or unlock the reel, depending on the state of the reel when clicked upon.

 OR

2. Select **Options > Locked** from the menu bar (if the menu-bar item is shaded and checked).

Archiving a Reel

Reels that are no longer in use can be archived. Hidden archived reels are not visible in the list on the "Reel Editor Main Screen -- Geometry Tab" on page 328, but the information is still available should it ever be needed again.

➢ To archive a reel

1. Select **Options > Archived** (so that a check appears beside **Archived**) from the menu bar (from either the Geometry or Info tab).

 The **Archived** icon appears in the lower right-hand portion of the diagram panel.

2. Save the reel to retain archival information.

3. Click **OK**.

➢ To view archived reels

1. From the Select a Reel dialog, click (to check) **Show Archived Reels**.

Printing Reel Data

The reel's configuration and approximate capacities can be printed. This data can be printed to a printer or to a PDF file for emailing or viewing by others.

The free Adobe Reader must be installed to view PDF files. It can be downloaded from Adobe's Web site at www.adobe.com

CTES does not recommend using any PDF printer other than the CTES PDF Printer 25. Doing so may result in unpredictable and undesirable results.

➢ To print reel data to a printer

1. Select **Print** from the toolbar.

 The Reel Print dialog appears.

2. (Optional) Enter a **Title** for the report.

3. (Optional) Change the **Logo** to appear on the report.

4. Select the printer. (Select **Change** to set printer properties.)

 Any printer connected to the computer, either directly or through a network, can be selected. All available printers are listed in the drop-down menu.

5. Click **Print**.

 The data is sent to the selected printer.

➢ To print to the Cerberus PDF generator

1. Click **Print**  on the toolbar.

 The Reel Print dialog appears.

2. (Optional) Enter a **Title** for the report.

3. Select **Change** associated with **Printer Options**; then select **CTES PDF Printer 25**.

 Other printer options may also be selected here.

4. Click **Print**.

 The Save As dialog appears.

5. Enter a name for the file.

 If necessary, navigate to the desired folder where the file will be saved.

 Ensure the Save as type is listed as PDF (*.pdf). Otherwise, the file might not be saved correctly and might not be readable.

6. Click **Save**.

 The file is saved to the selected location, with the desired name.

Configuring Reels

There are several steps to configuring a reel. The following links point to paragraphs that cover these topics in more detail.

- "Configuring Reel Geometry" on page 335.
- "Configuring Reel Type" on page 336.
- "Configuring Service Data" on page 337.
- "Estimating Reel Capacity" on page 337.

Configuring Reel Geometry

The reel geometry must be entered for calculations. Reel-Trak uses the core diameter and width to estimate how many wraps of tubing are on the reel, and the effective bending radius, for a given length of tubing on the reel. Flange diameter is only used in the capacity estimating utility.

Reel-Trak assumes there is enough flange to accommodate the specified string length. Unless the Watch Dog feature is on, no warnings are given if the reel capacity is insufficient to hold the specified string.

➢ To enter reel geometry

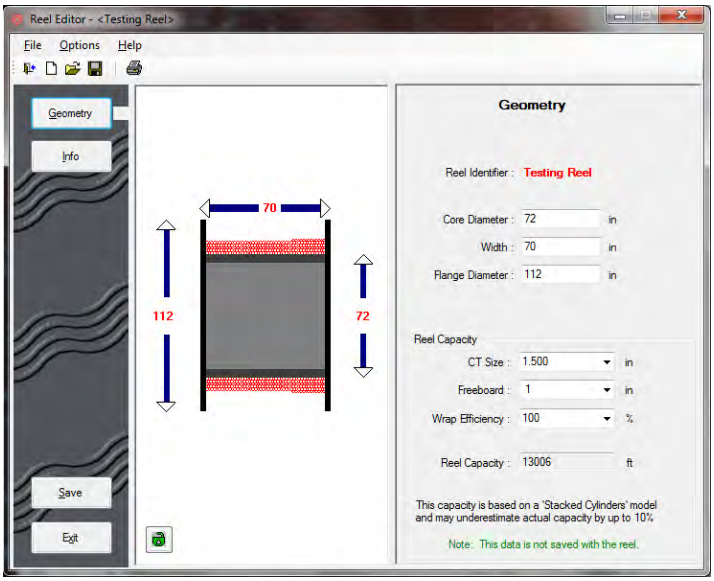
1. Click the **Geometry** tab.

2. Enter the reel geometry criteria within the Geometry panel.

 The reel criteria for the geometry include:

 - **Core Diameter** - The diameter of the core of the reel.
 - **Width** - The distance between the flanges, measured from the inside surface.
 - **Flange Diameter** - The diameter of the reel from the top to the bottom.

Geometry Panel in the Reel Editor Display Screen



Configuring Reel Type

The reel type is for record keeping purposes and is not used in any calculations. However, the reel type can be used to view or hide reels on the Reel Editor Main Screen -- Geometry Tab.

➢ To configure the reel type

1. Click the **Info** tab. The Information panel appears on the right-hand side of the Reel Editor display screen.

2. Select the type of reel.

 For a work reel, click the **Work Reel** radio button.

 For a shipping spool, click the **Shipping Spool** radio button.

3. For a work reel, select the following options.

- Select if the reel is a **Land Unit** or **Offshore Skid**.
- If the reel is e-line capable, select the **E-line Capable** checkbox.

4. Select the **Save** button on the tab panel or the **Save** toolbar icon.

Configuring Service Data

Service data can be entered for the reel for record keeping purposes. This data is not used in any calculations.

➢ To enter service data

1. Click the **Info** tab. The Information panel appears on the right-hand side of the Reel Editor display screen; the Service Data subheading appears within the Information panel.

2. Enter reel information for records.

 Record information includes the **Manufacturer** of the reel, the **Model** of the reel, the **Date Commissioned**, the **Empty Reel Weight**, and any **Comments** for the reel.

3. Select the **Save** button on the tab panel or the **Save**  toolbar icon.

4. Select the Exit button on the tab panel or the **Exit**  toolbar icon.

 The Service Data screen is closed and the user is returned to the main Reel Editor screen.

Estimating Reel Capacity

The reel capacity can be estimated for different diameters of CT. This utility is useful when a string is reversed onto a new reel or strings are spliced together (see page 258).



This estimate assumes perfect spooling and no "nestling" of one wrap into the one below it. In practice, up to 10% more tubing may fit on the reel than the estimate shown.

➢ To estimate reel capacity

1. From the Reel Editor display screen, select the Geometry tab.

2. Within the Reel Capacity section, select the **CT Size** from the dropdown menu.

 All available CT sizes are listed.

3. Select the amount of **Freeboard**.

 Freeboard is the distance, typically 2–4 inches, from the top wrap to the edge of the flange.

4. Select the **Wrap Efficiency.**

 The greater the wrap efficiency, the more CT can fit on the reel.

 The **Reel Capacity** calculates the estimated amount of CT that can fit on the selected reel, according to the information supplied.

Troubleshooting in Reel Editor

The following table lists some troubleshooting tips for Reel Editor.

Situation	Possible Cause	Action
I cannot make any changes to a reel.	The reel may be locked.	Try unlocking the reel. See “Locking a Reel” on page 333.
I don’t see a reel in the list that I know should be there.	The reel may be archived.	Try showing archived reels, work reels, and shipping spools on the Reel Editor screen. See “The Reel Editor Screen” on page 328.

Chapter 15
Using Fluid Manager

Fluid Manager is used to define the properties of fluids. Edit the standard fluids provided and create new fluids with Fluid Manager. Fluid Manager is accessible across the software suite, providing a consistent interface between applications.

After configuration, use fluids for calculations in Hydra and Hydra. The reservoir fluid in Well Manager can be chosen from the fluids configured in Fluid Manager. The following links point to paragraphs that cover these topics in greater detail.

- "Starting Fluid Manager" on page 339.
- "Types of Fluids" on page 340.
- "The Fluid Manager Display Screen" on page 339.
- "Working with Fluids" on page 343.
- "Calculating Properties for Liquids" on page 346.
- "Fluid Correction Factors" on page 347.
- "Troubleshooting in Fluid Manager" on page 348.

Starting Fluid Manager

➢ To start Fluid Manager

1. Find the Cerberus main menu (see page 21).
2. Select **Managers** > **Fluid Manager...** from the menu bar.

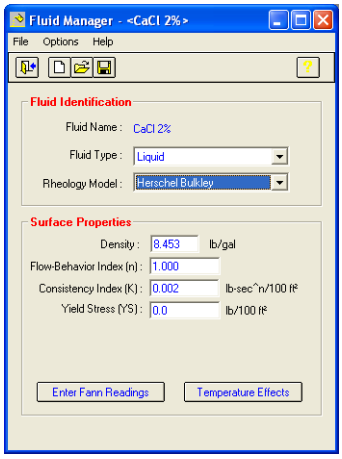
Launch Fluid Manager from other applications in the Cerberus suite by clicking **Fluid Manager** .

The Fluid Manager Display Screen

The Fluid Manager main screen contains the following areas:

- **Toolbar -** Allows for quick access to common tasks. For a full description of each button, see “Using the Fluid Manager Toolbar” on page 340.
- **Fluid Identification -** Look here to see the fluid name and select its type.
- **Fluid Data -** Enter the properties of the fluid here. The properties depend on the type of fluid being configured.

Fluid Manager main screen



Using the Fluid Manager Toolbar

Below is a description of the buttons on the main Fluid Manager toolbar, and each button's function.

Fluid Manager Main Toolbar

Tool Button	Tool Name	Function
	Exit	Exits Fluid Manager.
	New	Creates a new fluid. See “Creating a New Fluid” on page 343.
	Open	Opens an existing fluid. See “Opening an Existing Fluid” on page 343.
	Save	Saves the current fluid to disk.

Types of Fluids

There are four different types of fluids. Each type of fluid requires different inputs. The following links point to paragraphs that cover these topics in detail.

- "Liquid" on page 341.
- "Gas" on page 341.
- "Foam" on page 341.
- "Nodal (Multiphase) Fluid" on page 341.

Liquid

Liquids are divided into four different rheology models. Each of these are explained on the following pages:

- "Newtonian Fluid" on page 342.
- "Bingham Plastic" on page 342.
- "Power Law Fluid" on page 342.
- "Herschel-Bulkley" on page 343.

Gas

Fluid manager can currently define three types of gases: nitrogen, air, and natural gas.

The specific gravity of a gas is its molecular weight divided by the molecular weight of air (28.97 g/mol). The specific gravity must be input at standard conditions of 60º F and 14.7 psi. Since a gas is compressible, the density of the gas at downhole conditions of temperature and pressure will be calculated at run time.

Foam

A foam is a combination of a gas and base fluid. Despite being a multiphase mixture of a gas (usually nitrogen) and a liquid, a foam can often be modeled as a homogeneous (single) fluid with properties based on a combination of Power Law and Bingham Plastic models.

Hydra currently supports only nitrogen-based foams. The base liquid (the most common being water or a gel) must be configured as a separate fluid before configuring the foam. See "Configuring a Foam" on page 345. The foam quality is calculated at run time.

Nodal (Multiphase) Fluid

A nodal fluid is a combination of oil, gas, and water. The gas can be nitrogen, air, or natural gas. The term "nodal" refers to the mathematical methods used to calculate the pressure drop of such a mixture. Nodal fluids are most commonly encountered in the following cases:

- As the reservoir fluid, specified in Well Manager.
- When the reservoir is producing fluid and a different fluid is being pumped down the CT. Hydra treats the mingled fluids in the annulus as nodal.
- As a commingled mixture of nitrogen and liquid used for stimulation or well cleanout. In this case, the water fraction should be set to 1 (one), indicating no oil is present in the liquid phase.



The components of the nodal fluid must be specified at standard conditions of 60º F and 14.7 psi.

Newtonian Fluid

Newtonian fluid is the model for a fluid that flows in a linear manner. A Newtonian fluid is represented by the following equation:

$$shear\ stress = viscosity * shear\ rate$$

Few oil field fluids, other than water and water-like fluids, are truly Newtonian, but the Newtonian model is the best known and easiest to understand. The Newtonian model provides the basis of several important methods used to calculate friction factors and pressure drops.

Fresh water can be modeled as a Newtonian fluid.

The density and viscosity for a Newtonian fluid will need to be known. The viscosity can be calculated from the fluid's viscometer readings. See “Calculating Properties for Liquids” on page 346.

Bingham Plastic

A Bingham Plastic fluid does not flow until a specific yield point is reached. A Bingham Plastic fluid is represented by the following equation:

$$shear\ stress = yield\ point + (plastic\ viscosity * shear\ rate)$$

Cement slurry and some drilling muds are often modeled as Bingham Plastics.

For a Bingham Plastic fluid, its density, plastic viscosity (PV), and yield point (YP) need to be known. The PV and YP can be calculated from the fluid's viscometer readings. See “Calculating Properties for Liquids” on page 346.

Power Law Fluid

A Power Law fluid does not behave in a linear manner. However, unlike a Bingham Plastic fluid, a Power Law fluid does not require a minimum yield stress to start flowing. A Power Law fluid is represented by the following equation:

$$shear\ stress = consistency\ factor * (shear\ rate^{\ behavior\ index})$$

Many oil field fluids, such as water-based muds, gels, and hydrocarbons, are modeled using the Power Law model.

For a Power Law fluid, its density, consistency factor (K'), and behavior index (n') should be known. K' and n' can be calculated from the fluid's viscometer readings. See “Calculating Properties for Liquids” on page 346.

Herschel-Bulkley

The Herschel-Bulkley model, a three-parameter rheological model, combines the features of both the Power Law and Bingham Plastic models. It allows a yield stress with Power Law behavior at higher stress levels. It is represented by the following equation:

$$shear\ stress = yield\ stress + consistency\ factor * (shear\ rate^{behavior\ index})$$

The Herschel-Bulkley model is often preferred to Power Law or Bingham Plastic relationships because it results in more accurate models of rheological behaviors when adequate data are available.

For a Herschel-Bulkley model, its density, consistency factor (K'), yield point (YP), and behavior index (n') need to be known. The n', K', and YP can be calculated from the fluid's viscometer readings. See “Calculating Properties for Liquids” on page 346.

Working with Fluids

This section deals with working with fluids.

Creating a New Fluid

Fluid Manager comes with some common fluids already defined. New fluids can be created.

To model nitrified water, create a nodal (multi-phase) fluid and set the water fraction to 1 (one) and the gas type to nitrogen.

➢ To create a new fluid

1. Click **New** on the Fluid Manager toolbar.
2. Enter a name for the new fluid.
3. Click **OK**.
4. Select the **Fluid Type** from the dropdown menu.
5. For liquids or foams, select the **Rheology Model** from the dropdown menu.
6. Enter the properties for the fluid, according to the fluid type.

 See “Types of Fluids” on page 340.

7. Click **Save** on the toolbar to save the new fluid.

Opening an Existing Fluid

When a fluid has been configured, it is available for use in other applications in Cerberus. There are two ways to open an existing fluid:

- Click **Open** on the toolbar.

 OR

- Select **File** > **Open** from the menu bar.

Saving a Fluid

Saving a fluid records changes to the fluid on the hard drive. There are three ways to save a fluid:

- Click **Save** on the toolbar.

 OR

- Select **File** > **Save** from the menu bar.

 OR

- Select **File** > **Save as...** from the menu bar.

Deleting a Fluid

Unnecessary fluids can be deleted. However, use caution when deleting fluids used in projects. When a fluid is deleted, the information is not recoverable and must be recreated if the fluid is needed at a later time.

A fluid used in a project, or used as a reservoir fluid in Well Manager, should not be deleted. If a project referencing a deleted fluid is opened, an error is generated.

➢ To delete a fluid

1. **Open** the fluid in Fluid Manager.
2. Select **File** > **Delete** from the menu bar.
3. Click **Yes** to confirm deleting the fluid.

Importing a Fluid

Share fluids between computers by importing and exporting them.

➢ To import a fluid

1. If importing from removable media, **insert** the media in the drive.
2. **Start** Fluid Manager.
3. Select **File** > **Import Fluid** from the menu bar.
4. Select the drive and path where the fluid is located.
5. Select the fluid or fluids to import.

6. Click **Open**.
7. Click **Yes** to open the fluid.

To . . .	Do this . . .
Select a group of items that are next to each other.	Click the first item, then hold down the SHIFT key and click the last item.
Select multiple non-adjacent items.	Hold down the CTRL key and click each item.

Exporting a Fluid

Export a fluid for transfer to another computer.

➢ To export a fluid

1. If exporting to removable media, insert the media in the drive.
2. Start Fluid Manager.
3. Select **File** > **Export Fluid...** from the menu bar.
4. Select the fluid(s) to download.
5. Select the destination drive and path.
6. Click **Save**.
7. Click **OK** to return to the main display screen.

To . . .	Do this . . .
Select a group of items that are next to each other.	Click the first item, then hold down the SHIFT key and click the last item.
Select multiple non-adjacent items.	Hold down the CTRL key and click each item.

Configuring a Foam

A foam is a base liquid combined with a gas. Before configuring a foam, configure the base liquid.

➢ To create a foam

1. Create and configure the base liquid (the most common being water or a gel) as a separate fluid in Fluid Manager.
2. Create a new fluid.
3. Select Foam as the **Fluid Type**.
4. Choose the **Rheology Model** to use.
5. Select the newly configured base liquid as the **Base Fluid**.

Calculating Properties for Liquids

The user may calculate fluid properties for Newtonian fluids, Power Law fluids, Bingham Plastics, and Herschel-Bulkley fluids from their Fann35 viscometer readings. In addition, the effect of temperature on the liquid viscosity can be estimated. See "Newtonian Fluid" on page 342, "Power Law Fluid" on page 342, "Bingham Plastic" on page 342, and "Herschel-Bulkley" on page 343 for more information.

➢ To calculate liquid properties at ambient / surface conditions

1. Open the fluid.
2. Choose which rheology model to use from the dropdown menu.
3. Click the **Enter Fann Readings** button.
4. Enter the viscometer readings in the appropriate fields.
5. Click the **Calculate** button to:
 - Calculate Viscosity for Newtonian fluids.
 - Calculate n and k for Power Law fluids.
 - Calculate PV and YP for Bingham Plastics.
 - Calculate n, k & YP for Herschel-Bulkley fluids.
6. To use the calculated values, click **Use**. Otherwise, click **Cancel**.



The standard R1-B1 rotor-bob combination and F1 torsion spring are assumed for all calculations.

If temperature effects on liquid viscosity are to be considered in Hydra calculations, complete the following procedure. Otherwise, save and exit Fluid Manager.

➢ To calculate temperature effects on liquid viscosity

1. Open the fluid.
2. Choose the rheology model to use from the dropdown menu.

3. Estimate the temperature effects on liquid viscosity by clicking on the **Temperature Effects** button.
4. Enter the rheology data at different temperatures.
5. Click **OK** to exit.
6. **Save** the fluid and exit Fluid Manager. If this fluid is loaded into Hydra and the **Temperature Effects...** option is enabled, the values of the constants are used in the Hydra calculations.

Fluid Correction Factors

If model predictions for a particular fluid differ from observed readings by a consistent amount, a correction factor can be applied to the fluid to compensate. This fluid correction factor applies to Newtonian, Power Law, Bingham Plastic, Herschel-Bulkley, and Foam fluids. See the following paragraphs for more information:

- "Entering Fluid Correction Factors" on page 347.
- "Using Fluid Correction Factors" on page 347.

Entering Fluid Correction Factors

The following procedure outlines how to enter a Fluid Correction Factor.

➢ To enter a correction factor

1. Open the fluid.
2. Select **Options** > **Correction Factor** from the menu bar.
3. Enter the correction factor and click **OK**.
4. Save the fluid by clicking **Save**  on the toolbar.

Using Fluid Correction Factors

The following procedure outlines how to use a Fluid Correction Factor.

➢ To use correction factors in Hydra

1. Open the project in Hydra.
2. Click the **Project Options**  button.
3. Click the **General** tab.
4. Select the **Include fluid correction factors** checkbox.
5. Click **OK** to close the **Options** window.

Troubleshooting in Fluid Manager

Situation	Possible Cause	Action
I don't see the base fluid I want to use for my foam.	Fluid Manager expects the base fluid to be configured before the foam.	Configure the base fluid as a separate fluid, then configure the foam.

Chapter 16
Using Equipment Manager

Equipment Manager is used to configure the information about different pieces of surface equipment involved in CT jobs. Equipment Manager stores all its equipment in a single database. Individual pieces of equipment can be grouped into equipment packages for an individual job.

- "Starting Equipment Manager" on page 349.
- "The Equipment Manager Screen" on page 349.
- "Working with Pieces of Equipment" on page 351.
- "Creating an Equipment Package" on page 353.

Starting Equipment Manager

➢ To Start Equipment Manager

1. Find the Cerberus main menu or home page (see page 21).
2. Select **Equipment Manager** from the **Managers** menu.
3. Equipment Manager will now start.

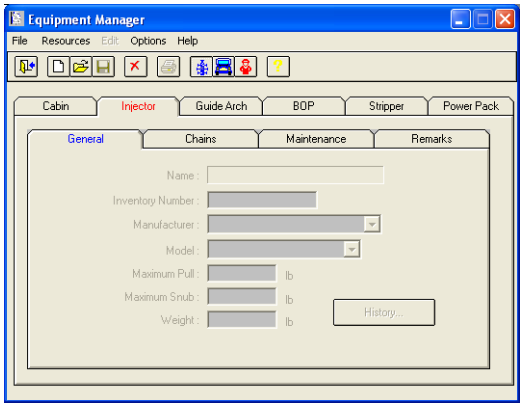
The Equipment Manager Screen

The Equipment Manager Main Screen has the following areas:

- **Toolbar.** Click these buttons for quick access to common tasks. See "The Equipment Manager Toolbars" on page 350.
- **Cabin Tab.** Configure cabins here.
- **Injector Tab.** Configure injectors here.
- **Guide Arch Tab.** Configure guide arches here.
- **BOP Tab.** Configure BOPs here.

- **Stripper Tab.** Configure strippers here.
- **Power Pack Tab.** Configure power packs here.

Equipment Manager Main Screen



The Equipment Manager Toolbars

Rest the cursor over a tool button for a few seconds for the tool name to appear.

Below is a description of the buttons on the main Equipment Manager toolbar, and each button's function.

Equipment Manager Main Toolbar

Tool Button	Tool Name	Function
	Exit	Exits Equipment Manager.
	New	Opens the dialog box to create new equipment (See "Creating New Equipment" on page 352.) or a new equipment package (See "Creating an Equipment Package" on page 353.).
	Open	Opens the dialog box to open an existing piece of equipment or equipment package. See "Opening Existing Equipment" on page 352.
	Save	Saves the current piece of equipment or equipment package to disk. See "Saving Equipment" on page 352.
	Delete	Deletes the current piece of equipment or equipment package. See "Deleting Equipment" on page 353.

Equipment Manager Main Toolbar

Tool Button	Tool Name	Function
	Print	This option is currently disabled.
	Equipment	Opens the Equipment Manager
	Auxiliary	Opens the Vehicle tab for configuration.
	Personnel Manager	Opens the Personnel Manager.
	Help	Opens Help for Equipment Manager.

Working with Pieces of Equipment

This section deals with working with individual pieces of equipment. The following types of equipment can be configured:

- Cabins
- Injectors
- Guide Arches
- BOPs
- Strippers
- Power Packs
- Vehicles

This section includes the following topics:

- “Creating New Equipment” on page 352.
- “Opening Existing Equipment” on page 352.
- “Saving Equipment” on page 352.
- “Deleting Equipment” on page 353.

Creating New Equipment

All of the pieces of equipment in Equipment Manager are created in the same way. The only difference is the inputs for each type.

➢ To Create a Piece of Equipment

1. Start Equipment Manager.
2. Click the appropriate tab for the type of equipment to be created.
3. Click the **New** button  on the toolbar.
4. Enter a name for the piece of equipment and click **OK**.
5. Enter the inputs relevant to that piece of equipment.

Opening Existing Equipment

An existing piece of equipment can be opened in order to review or edit its inputs.

➢ To Open a Piece of Equipment

1. Start Equipment Manager.
2. Click the appropriate tab for the type of equipment to open.
3. Click the **Open** button  on the toolbar.
4. Select the name of the piece of equipment and click **OK**.
5. The inputs for that piece of equipment can now be viewed and edited.

 *Changes to a piece of equipment should be saved before moving on to the next piece of equipment.*

Saving Equipment

Pieces of equipment are saved to record changes to the hard drive.

➢ Ways To Save a Equipment

- Click the **Save** button  on the toolbar.

 - Or -

- Select **File** > **Save** from the menu bar.

 *Changes to a piece of equipment should be saved before moving on to the next piece of equipment.*

Deleting Equipment

Unnecessary equipment no longer in use can be deleted.

➢ To Delete a Piece of Equipment

1. Open the piece of equipment in Equipment Manager.
2. Click the **Delete** button  on the toolbar.

Creating an Equipment Package

An equipment package is assembled in Equipment Manager. Then that equipment package can be selected for a job in Job Manager.

➢ To Create an Equipment Package

1. Start Equipment Manager.
2. Select **Resources** > **Equipment Package** from the menu bar.
3. Click the **New** button  on the toolbar.
4. Enter a name for the equipment package and click **OK**.
5. Select each of the individual components in the package.



Individual pieces of equipment should be saved before moving on to the next piece of equipment.

Chapter 17
Using Personnel Manager

Personnel Manager is used to configure the information about the different people who make up the crew on a CT job. Personnel Manager stores all its personnel records in a single database. Personnel can be selected for an individual job in Job Manager.

- "Starting Personnel Manager" on page 355.
- "The Personnel Manager Screen" on page 355.
- "Working with Personnel Records" on page 357.

Starting Personnel Manager

➢ To Start Personnel Manager

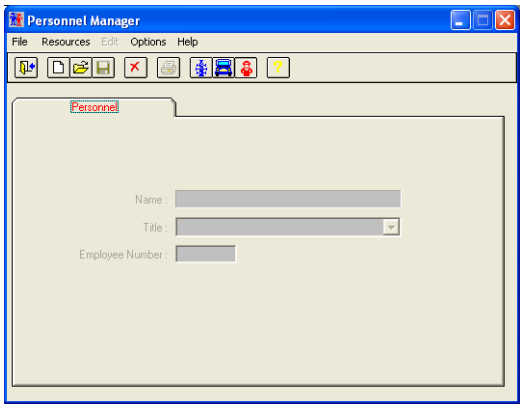
1. Find the Cerberus main menu or home page (see page 21).
2. Select **Personnel Manager** from the **Managers** menu.
3. Personnel Manager will now start.

The Personnel Manager Screen

The Personnel Manager main screen has the following areas:

- **Toolbar.** Click these buttons for quick access to common tasks. See "The Personnel Manager Toolbars" on page 356.
- **Personnel Tab.** Configure personnel here.

Personnel Manager main screen



The Personnel Manager Toolbars

Below is a description of the buttons on the main Personnel Manager toolbar, and each button's function.

Personnel Manager Main Toolbar

Tool Button	Tool Name	Function
	Exit	Exits Personnel Manager.
	New	Opens the dialog box to create a new personnel record. See "Creating a New Personnel Record" on page 357.
	Open	Opens the dialog box to open an existing personnel record. See "Opening an Existing Personnel Record" on page 357.
	Save	Saves the current personnel record to disk. See "Saving a Personnel Record" on page 358.
	Delete	Deletes the current personnel record. See "Deleting a Personnel Record" on page 358.
	Print	This option is currently disabled.
	Equipment	Opens the Equipment Manager
	Auxiliary	Opens the Vehicle tab for configuration.

Personnel Manager Main Toolbar

Tool Button	Tool Name	Function
	Personnel Manager	Opens the Personnel Manager.
	Help	Opens Help for Equipment Manager.

Working with Personnel Records

This section deals with working with personnel records. This section includes the following topics:

- "Creating a New Personnel Record" on page 357.
- "Opening an Existing Personnel Record" on page 357.
- "Saving a Personnel Record" on page 358.
- "Deleting a Personnel Record" on page 358.

Creating a New Personnel Record

All personnel records are configured in the same way.

➢ To Create a Personnel Record

1. Start Personnel Manager.
2. Click the **New** button  on the toolbar.
3. Enter a name for the person and click **OK**.
4. Enter the inputs relevant to that person.

Opening an Existing Personnel Record

A personnel record can be opened to review and edit its inputs.

➢ To Open a Personnel Record

1. Start Personnel Manager.
2. Click the **Open** button  on the toolbar.

3. Select the name of the personnel record and click **OK**.

4. The inputs for that record can now be view and edited.

Saving a Personnel Record

Personnel records are saved in order to store changes on the hard drive.

➢ Ways To Save a Personnel Record

- Click the **Save** button  on the toolbar.

 OR

- Select **File**, **Save** from the menu.

Deleting a Personnel Record

Unnecessary personnel records can be deleted.

➢ To Delete a Personnel Record

1. Start Personnel Manager.

2. Click the **Open** button  on the toolbar.

3. Select the name of the personnel record and click **OK**.

4. Click the **Delete** button  on the toolbar.

Chapter 18
Using Achilles

Achilles is the CTES fatigue model. Achilles is used in two different ways in Cerberus:

- Reel-Trak uses Achilles to calculate fatigue for each segment of a string. This use of Achilles in this case is invisible to the user.
- Achilles is used as a modeling tool for CT string design.

Using Achilles as a modeling tool for CT string design provides a means of determining the relative effect of the various factors influencing tubing fatigue life. For example, Achilles helps answer questions such as, "How does increasing the tubing size from 1.5" to 1.75" affect the expected life?" or "How much does using a 94" gooseneck instead of a 72" gooseneck improve tubing life?"

Achilles assumes that the CT is cycled for its entire life using consistent bending radius and pressure. While this is not physically realistic, it provides a means to compare and contrast the effect of key parameters and hence optimize the best configuration for an upcoming job or string purchase.

Starting Achilles

➢ To start Achilles

- From the Cerberus home page, select, from the menu bar, **Calculations > Achilles**.
- Launch Achilles from other applications in the Cerberus suite by clicking the **Achilles** button .

Fatigue Model Theory

Fatigue results from plastic deformation when the steel CT is bent off the reel and around the gooseneck. The amount of fatigue that a bending event causes depends primarily on the CT material, the CT geometry, the pressure, the bending radius, and the current fatigue. Achilles uses these factors to calculate the amount of fatigue created by a trip into and out of the hole. Because Achilles takes these factors into account, it provides a more accurate estimate of CT life than other methods of estimating fatigue, such as running feet.

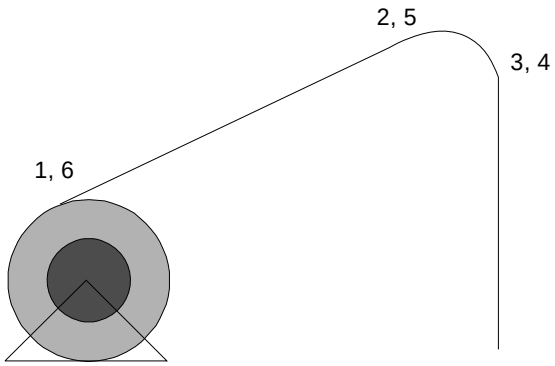
When CT accumulates too much fatigue, it fails. First, a microscopic crack begins to form in the wall of the CT. Eventually the crack propagates through the wall, and the CT cannot hold pressure. At high pressure, these two stages may seem simultaneous. Achilles can calculate the number of trips to the first stage – crack initiation, or the second stage – fracture. CTES recommends using CT to no more than 80 percent of its used life.

Achilles, along with the rest of Cerberus, assumes that the yield strength of CT is not affected by fatigue. However, there is some evidence that the yield strength of used CT is reduced by up to 15%.

Bending Events for CT

Achilles uses the six bending events listed below to calculate fatigue for each trip into and out of the hole.

1. Off the reel - bent to straight.
2. Onto the gooseneck (guide arch) - straight to bent.
3. Off the gooseneck to the injector - bent to straight.
4. From the injector onto the gooseneck - straight to bent.
5. Off the gooseneck toward the reel - bent to straight.
6. Onto the reel - straight to bent.



CT does not fatigue downhole because the radius of curvature of the wellbore trajectory is not small enough to cause plastic yielding of the CT (except in extraordinarily rare circumstances – rare enough to be considered negligible).

CT also does not fatigue when the CT doesn't move, even if the pressure changes, such as during a pressure test.

Fatigue Model Inputs

Achilles uses the following inputs to calculate fatigue for each bending event:

- CT Diameter
- CT Wall Thickness
- CT Material Type
- Current Fatigue (invisible to user)
- Bending Radius (at reel core or at guide arch)
- Pressure

How to Minimize Fatigue

When a string accumulates fatigue, the amount of fatigue on the string cannot be reduced. However, a new string can be designed to minimize fatigue, and to control the amount of new fatigue a string accumulates in the future.

➢ To minimize fatigue (new string)

- Use a thicker CT wall.
- Use a smaller outside diameter.
- Use a material appropriate for anticipated pressures.

➢ To minimize fatigue (used string)

- Use a larger guide arch.
- Perform fewer cycles.
- Cycle with lower pump pressure.
- Place the string on reel with a larger core, or use a reel core expander.
- Cut the string to shift new fatigue away from areas with current high fatigue, welds, and other areas of concern (zones).

Using Achilles

The Achilles Interface

Viewable within the main display screen are the different factors affecting fatigue life and diameter growth. Achilles models fatigue in terms of trips. Each trip is calculated according to the current configuration displayed on the screen. Thus, if the calculations are paused, a parameter is changed, and the calculations are

resumed, the model uses the revised configuration. Changing the diameter or wall size is physically meaningless. However, the pressure or bending geometry can be changed to reflect possible field conditions.

Three tabs are initially available from the Achilles interface: **Inputs**, **Analyzer**, and **Options**. A fourth tab, **Analyzer Data**, becomes available only when the **Calculate** button is selected on the **Analyzer** tab.

Before the user begins to run fatigue analyses, the parameters in the Options tab should be evaluated/set. See "Using the Options Tab" on page 367 for more information.

The Achilles Toolbar

The Achilles Main Toolbar

Tool Button	Tool Name	Function
	Exit	Exits Achilles.
	Print	Accesses the Achilles Print dialog to print data.

Using the Inputs Tab

The **Inputs** tab allows the user to vary each of the potential fatigue-affecting parameters and to run the fatigue calculation in a VCR-like environment; in other words, to run, step, etc., through the fatigue calculation.

➢ To run the coiled tubing fatigue calculation

1. Input the tubing dimensions (using dropdowns where available) **CT Material**, **CT Nominal Diameter**, **CT Nominal Wall**, **Pressure**, **Reel Diameter**, and **Arch Radius**.

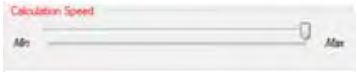
 Click the **Edit** to add/delete values that pertain to the Gooseneck **Arch Radius**. Values below 25 inches are not allowed.

2. Select **Calculation Speed** (**Min**/**Max**) to determine how fast the calculation runs.

3. Click **Play**  **, Step** (to view one trip at a time)  , **Stop**  , or **Reset**  to observe the fatigue calculation process.

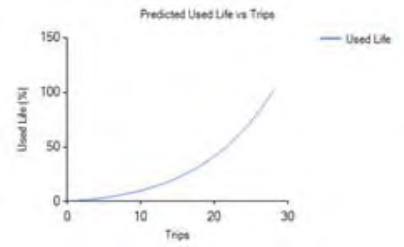
Rest the cursor over a tool button for a few seconds for the tool name to appear.

a. To adjust the **Calculation Speed**, click and drag the mouse anywhere along the **Min/Max** line.



4. The resulting data is displayed graphically in the right-hand portion of the **Inputs** tab.

a. The graph shown in the upper right-hand quadrant appears similarly to the following:



Two views of the graph are available: fatigue prediction or diameter prediction.

➢ To switch between type of graph viewed

Click (to toggle) either **Fatigue Prediction** or **Diameter Prediction** underneath the graph area of the Achilles Inputs-tab display screen.

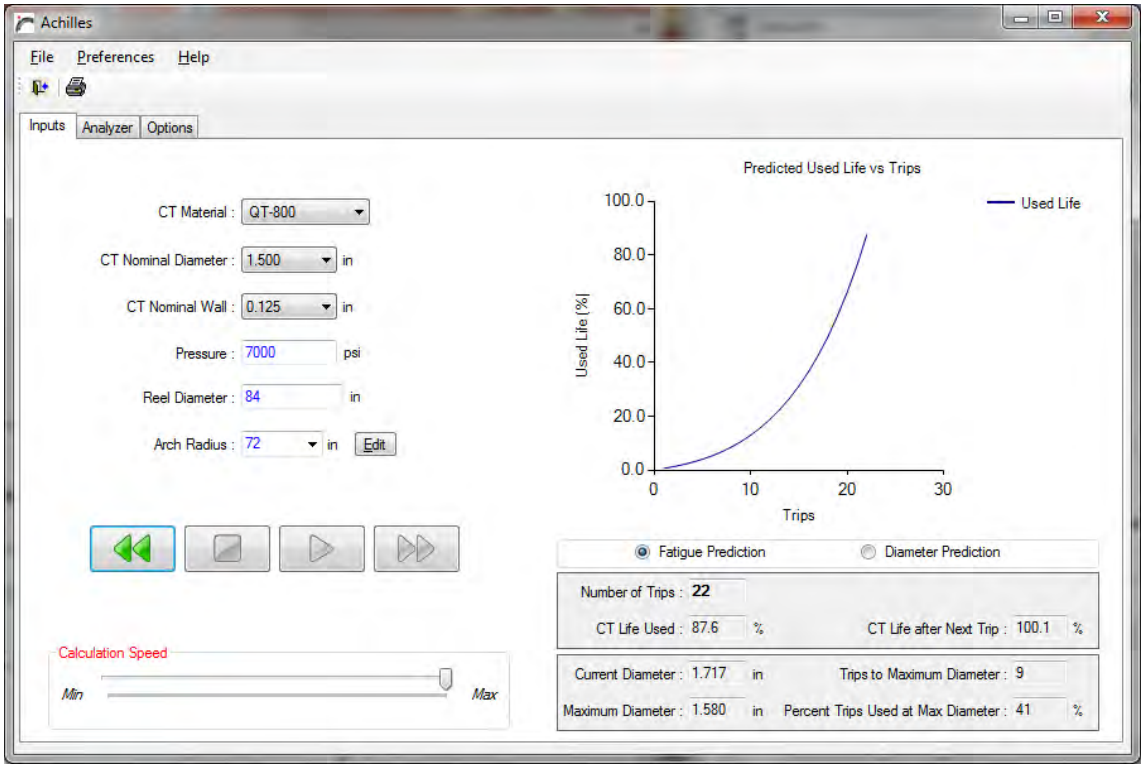
b. The **Fatigue Life Results** fields include:

- **CT Life Used**
- **Number of Trips**
- **CT Life after next trip**: Though the user may expect the program to stop when exactly 100% of the CT life is used, the last trip in/out of the well may cause **CT Life Used** to exceed 100%. If this is the case (and if the calculation cannot exceed 100% fatigue per the Option selected), the program stops before 100% and shows the user (**CT Life after next trip**) what the life will/would be after the next trip.

c. The **Diameter Growth Results** fields include:

- Current Diameter
- Maximum Diameter
- Trips to Maximum Diameter
- Percent Trips Used at Maximum Diameter

Achilles Inputs Tab



Using the Analyzer Tab

The Achilles Analyzer is used to model the comparison of different input values over a pressure range. The Analyzer compares the input parameters and their values. Then, the Analyzer produces a line-graph of trips to failure vs. pressure with lines for the different values.

The number of calculations depends on the pressure range and the number of values to compare. The greater the pressure range, the smaller the steps.

The larger the number of values, the more calculations the Analyzer must perform. The number of calculations and the current calculation are shown on the bottom right-hand portion of the screen.

➢ To use the Achilles Analyzer

1. Change the **Pressure** by entering the new range (**From/To**) and **Step** size.
2. Enter/select values for the **Fixed Variables** within the left-hand portion of the display screen.
3. Select the **Floating Variables**:
 a. Select the **Parameter** from the dropdown.
 b. Select **Value**(s) from the dropdown.

 OR

Input **Value**(s) and click the **Add** button (which appears when the user starts typing into the **Value** field) to include them in the list.

4. Click the **Calculate** button.

5. Achilles performs the calculations for the different cases. When Achilles is done calculating, a line graph appears.
 - A line is available for each value chosen for the floating variable.
 - Various CT and calculation information is found in the lower right-hand corner of the interface.

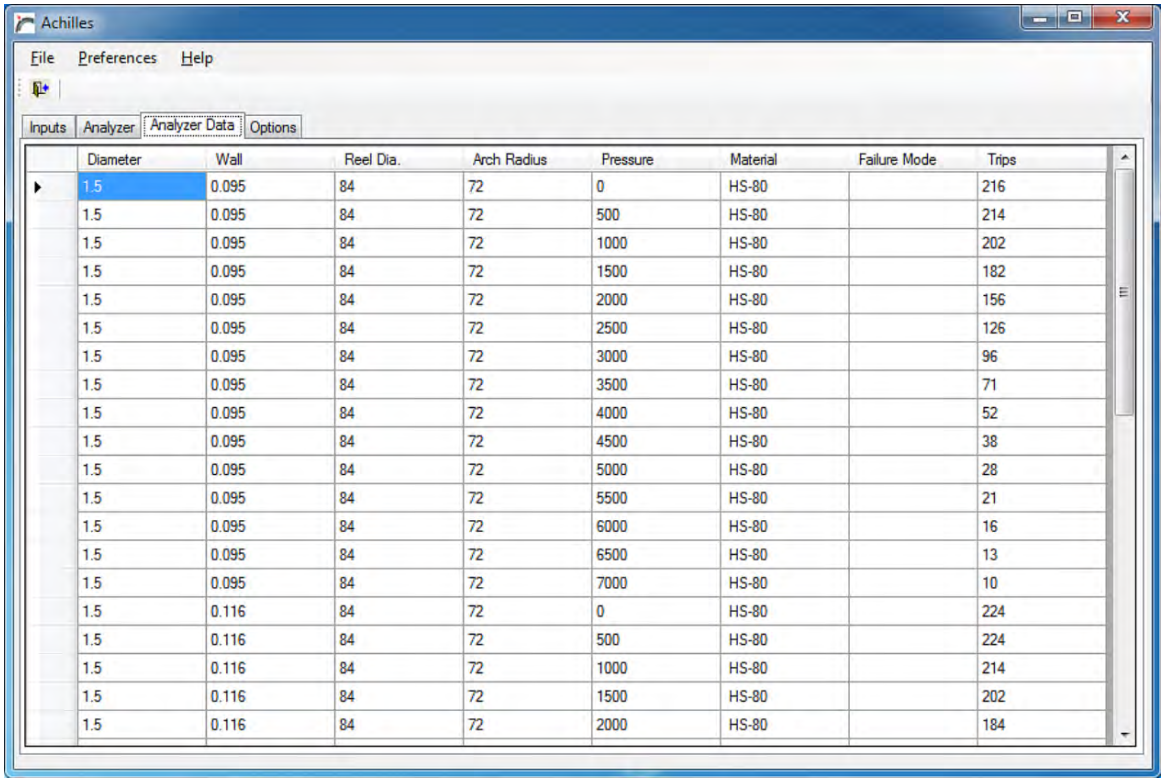
Achilles Analyzer Tab



Using the Analyzer Data Tab

After the fatigue calculation is run from the **Analyzer** tab, the **Analyzer Data** tab becomes available. This tab provides data for each **Pressure Step** chosen by the user.

Achilles Analyzer Data Tab



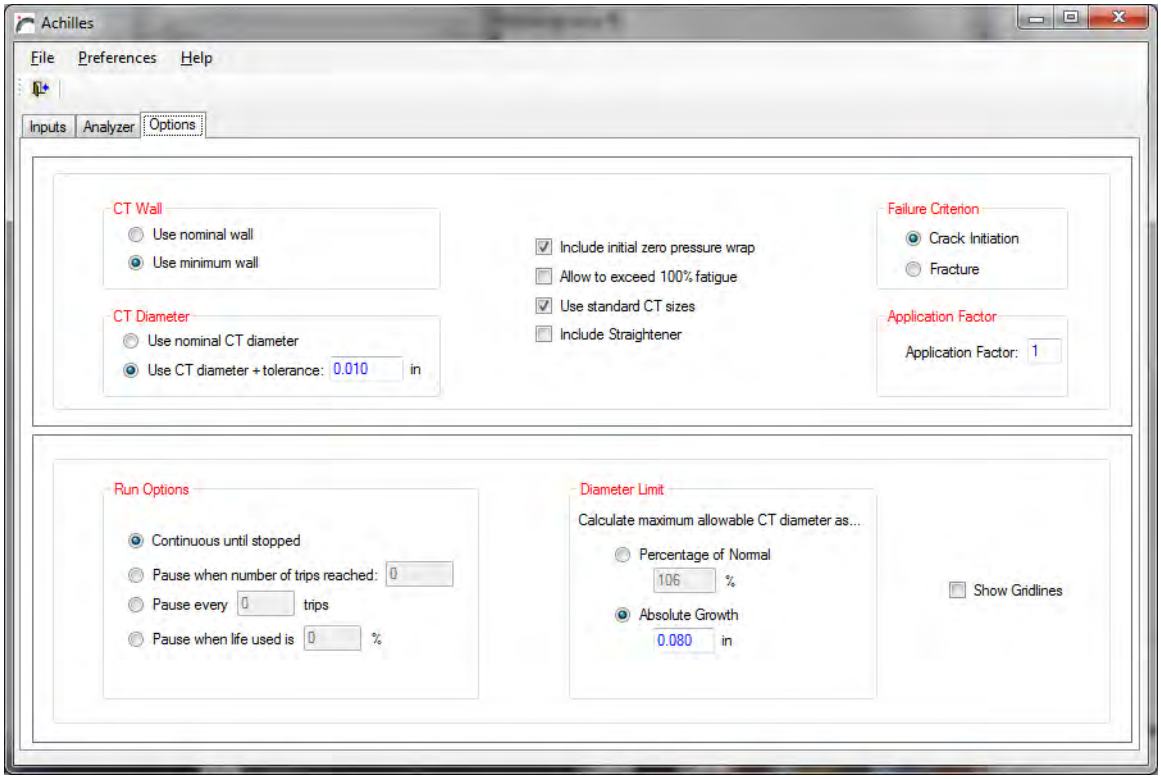
Diameter	Wall	Reel Dia.	Arch Radius	Pressure	Material	Failure Mode	Trips
1.5	0.095	84	72	0	HS-80		216
1.5	0.095	84	72	500	HS-80		214
1.5	0.095	84	72	1000	HS-80		202
1.5	0.095	84	72	1500	HS-80		182
1.5	0.095	84	72	2000	HS-80		156
1.5	0.095	84	72	2500	HS-80		126
1.5	0.095	84	72	3000	HS-80		96
1.5	0.095	84	72	3500	HS-80		71
1.5	0.095	84	72	4000	HS-80		52
1.5	0.095	84	72	4500	HS-80		38
1.5	0.095	84	72	5000	HS-80		28
1.5	0.095	84	72	5500	HS-80		21
1.5	0.095	84	72	6000	HS-80		16
1.5	0.095	84	72	6500	HS-80		13
1.5	0.095	84	72	7000	HS-80		10
1.5	0.116	84	72	0	HS-80		224
1.5	0.116	84	72	500	HS-80		224
1.5	0.116	84	72	1000	HS-80		214
1.5	0.116	84	72	1500	HS-80		202
1.5	0.116	84	72	2000	HS-80		184

Click a heading to sort the table by that field.

Using the Options Tab

Before the user begins to run any fatigue analyses, the parameters in the Options tab should be evaluated/set. Option selection causes different values to be applied to the fatigue calculations. Select the options that closely match the desired model.

Achilles Options Tab



Chapter 19
Using Hercules

Hercules is the coiled tubing limits model, used to calculate the burst and collapse pressures, and maximum tensile and compressive axial loads on tubing. All limits are graphed to determine the pressures and axial loads to use. The following links point to paragraphs that cover these topics in greater detail.

- "Starting Hercules" on page 369.
- "Tubing Limits Theory" on page 369.
- "The Hercules Display Screen" on page 373.
- "Working with Hercules Projects" on page 374.
- "Calculating Results in Hercules" on page 375.
- "Troubleshooting in Hercules" on page 377.

Starting Hercules

➢ To start Hercules

- From the Cerberus main menu, select **Calculations > Hercules** from the menu bar.
- Launch Hercules from other applications in the Cerberus suite by clicking the **Hercules** button .

Tubing Limits Theory

There are two components to the tubing limits theory: the stresses that create the limits, and the curves that represent the limits. The following links point to paragraphs that cover these topics in greater detail.

- "Coiled Tubing Stresses" on page 369.
- "Limit Curves" on page 371.

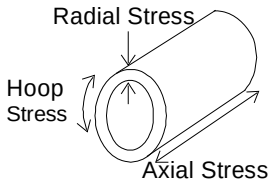
Coiled Tubing Stresses

Coiled tubing (CT) stresses are caused by:

- internal pressure
- external pressure
- axial force (tension or compression)
- torque

Combining internal / external pressure, and axial force with the geometry of the CT and the well, creates the following three principle stresses:

- Axial Stress
- Radial Stress
- Hoop Stress



Torque creates an associated shear stress.

The three principle stresses and the shear stress are combined using the von Mises Yield Condition to determine what pressure and axial force combinations will cause the CT to yield.

Axial Stress

Axial stress is caused by the axial force (tension or compression) applied to CT. If the compressive force exceeds the helical buckling load, the CT forms a helix in the hole. This helix causes an additional axial bending stress in the CT. (See “Helical Buckling” on page 390.)

When the CT is not helically buckled, the axial stress is the axial force divided by the cross-sectional area.

When the CT is in compression and helically buckled, the additional bending stress should be added to the axial stress. It is conservative to assume that the CT is buckled if the effective force is compressive (less than zero). Thus, the Pipe Limit Modeling Calculator assumes that the CT is buckled if there is any compressive effective force. Including helical buckling truncates the left side of the elliptical shape of the limits curve.

Radial Stress

The radial stress at a given location in the CT wall is the stress through the CT wall due to inner and outer pressures. The maximum stress always occurs at the inner or the outer surface. Since yielding occurs first at the inner surface, Hercules uses the radial stress at the inner surface in its calculations. The radial stress, simplified for the inner surface, is the negative of the internal pressure.

Hoop Stress

The hoop stress (also known as tangential stress) at a given location in the CT wall is the stress around the circumference of the CT due to inner and outer pressures. The maximum stress always occurs at the inner or the outer surface. Since yielding occurs first at the inner surface, Hercules uses the hoop stress at the inner surface in its calculations.

The Lame thick wall equation is used to calculate the hoop stress which involves a combination of the inner and outer pressures, and the squares of the inner and outer radii of the CT.

Torque

Torque is the moment of any force applied to the CT which causes or prevents rotation about the center axis. The result is torsion of the CT and an associated shear stress. The shear stress is greatest at the outer surface, and so this value is used as a worst case.

von Mises Yield Condition

The initial yield limit is based on the combination of the three principle stresses (Axial Stress, Radial Stress, and Hoop Stress) and Torque. The von Mises yield condition is commonly used to describe the yielding of steel under combined states of stress. In its general form, it includes the influence of all three principle stresses and shear stress, and involves the square of the differences between each of the principle stresses.

Limit Curves

The calculated graph takes into account maximum diameter considerations ("Maximum Diameter Considerations" on page 371) as well as safety factors ("Applying Safety Factors" on page 372). Also, see "How the Differential Pressures Curve is Calculated" on page 371.

Maximum Diameter Considerations

CT has a tendency to increase in diameter during its life. This change in geometry changes the stresses, and thus the limits. Hercules takes into account the diameter growth by calculating limits curves for both nominal and maximum diameters and selecting the most conservative portions.

How the Differential Pressures Curve is Calculated

The limits curve is determined by the stresses in the CT caused by inner pressure, outer pressure, and axial force. To simplify the presentation of the limits, the difference between the inner and outer pressure is defined as inner pressure minus outer pressure. However, the yield condition still remains a three variable equation. To represent the limits as a two dimensional curve, one of the variables must be held constant.

By holding the outer pressure constant and varying the axial force, inner pressure can be calculated, and the pressure difference can be plotted versus axial force. The resulting plot is an ellipse. Points on the graph where inner pressure was calculated to be negative are called imaginary data. (See "What is Imaginary Data?" on page 372.) Holding inner pressure constant instead of outer pressure produces a similar ellipse.

The differential pressure limits curve seen in Hercules is a conservative combination of four limits curves. Hercules calculates the limits curve for the four cases listed below. Hercules then selects the portions of these curves that are the most conservative (closest to the origin), producing the final limits curve.

Case	Diameter	Pressure*
1	nominal	0
2	maximum	0

Case	Diameter	Pressure*
3	nominal	maximum
4	maximum	maximum

*For the upper arch of the curve, Hercules uses a constant outer pressure. For the lower arch of the curve, Hercules uses a constant inner pressure.

How Constant Pressures Curves are Calculated

When Hercules calculates constant pressures limits curves, two half-ellipses are seen: one for the constant inner pressure, and one for the constant outer pressure. They are independent of each other and are calculated separately.

Each constant pressure limits curve seen in Hercules is a conservative combination of two limits curves. Hercules calculates the limit curve for the constant inner pressure at nominal diameter and maximum diameter. Hercules then selects the portions of these two curves that are the most conservative (closest to the origin), producing the final limits curve. Hercules uses the same process for the constant outer pressure. The user specifies maximum diameter and the constant inner pressure and constant outer pressure.

What is Imaginary Data?

Imaginary data results from calculations that produce negative pressures. Although negative pressures do not exist in reality, they can be calculated and graphed.

For example, the y-axis of the limits curve for the differential pressures graph represents the pressure difference (inner pressure minus outer pressure). Above the x-axis inner pressure is greater. Below the x-axis outer pressure is greater. If the limits curve is calculated with an outer pressure of zero, part of the curve falls below the x-axis. However, below the x-axis, inner pressure must be less than outer pressure, which is zero! A similar situation arises when the inner pressure is held at zero. These points with negative pressures are called imaginary data.

Imaginary data actually has only a small impact on the graph. Although imaginary data does not exist in reality, it can be slightly more conservative. Including imaginary data in the limits curve is a personal preference.

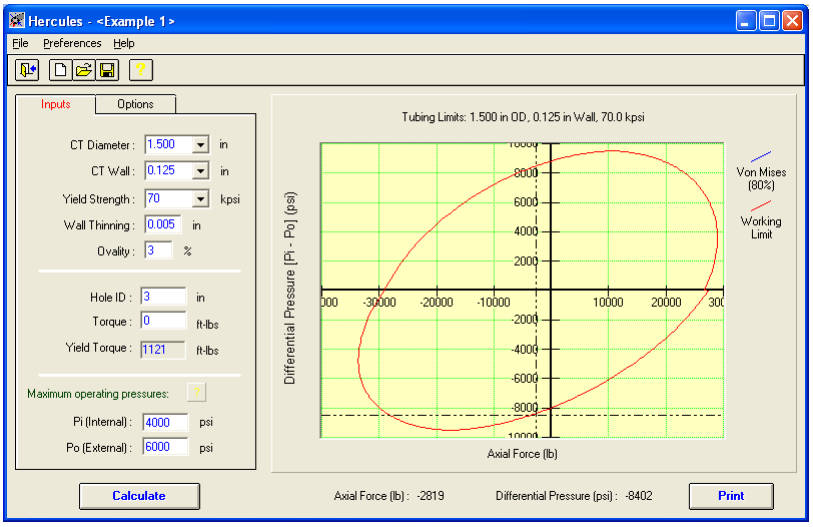
Applying Safety Factors

The limits curve calculated by the Von Mises yield condition represents where the CT would begin to yield. Although combining multiple limits curves is conservative, a larger buffer area is desirable. Hercules introduces more safety by multiplying the limits curve by safety factors to obtain a working limits curve.

Hercules provides for using different safety factors for burst and for collapse. The safety factor for collapse should be more conservative to account for ovality if one of the two collapse algorithms provided is not used. Both the API 5C7 and Plastic collapse models use ovality as an input. During its life the CT becomes somewhat oval due to bending on the reel and over the gooseneck. (Some ovality is removed by the gripper blocks on the chains.) Increased ovality increases the likelihood of collapse.

The Hercules Display Screen

Hercules Display Screen



The Hercules Toolbar

Rest the cursor over a tool button for a few seconds for the tool name to appear

Hercules Main Toolbar

Tool Button	Tool Name	Function
	Exit	Exits Hercules.
	New	Starts a new Hercules project. See “Creating a New Hercules Project” on page 374.
	Open	Opens the dialog box to open an existing Hercules project. See “Opening an Existing Hercules Project” on page 374.
	Save	Saves the current Hercules project.

Working with Hercules Projects

Hercules is organized by projects. A configuration in a project is stored so that it is accessible without entering all of the inputs again. All data is stored in a single database. The following links point to paragraphs that cover these topics in greater detail.

- "Creating a New Hercules Project" on page 374.
- "Opening an Existing Hercules Project" on page 374.
- "Saving a Hercules Project" on page 374.
- "Deleting a Hercules Project" on page 375.

Creating a New Hercules Project

The following procedure outlines how to create a new Hercules project.

➢ To create a new project

1. Click **New**  on the toolbar.
2. Enter a **project name**.
3. Click **OK**.

 Limits can now be calculated. See "Calculating Results in Hercules" on page 375.

Opening an Existing Hercules Project

The following procedure outlines how to open an existing Hercules project.

➢ To open an existing project

1. Click **Open**  on the toolbar.
2. Click the **name** of the project.
3. Click **OK**.

Saving a Hercules Project

When a project is saved in the Hercules, the configuration information is saved as well. When the project is reopened, the data is recalculated. There are two ways to save a project:

- Click **Save**  on the toolbar.

 OR
- Click **File** and select **Save** from the menu.

Deleting a Hercules Project

Unnecessary projects can be deleted. Deleted information cannot be recovered.

➢ To Delete a Project

1. Open the **project** to delete.
2. Select **File** > **Delete**.

Calculating Results in Hercules

Calculating the limits produces the limits curve for the given inputs. The following links point to paragraphs that cover these topics in greater detail.

- "Calculating Limits" on page 375.
- "The Differential Pressures Curve" on page 376.
- "Reading the Differential Pressures Graph" on page 376.

Calculating Limits

Before working with limits, they must be calculated. This procedure outlines how to calculate the limits.

➢ To calculate limits

1. Open the **project** in Hercules.
2. Enter parameters under the Inputs tab:

 The following parameter categories are needed:

 - CT geometry
 - Hole ID of the well
 - Torque
 - Maximum operating pressures
3. Select calculation options.

 The following calculation options are available under the Options tab:

 - Helical buckling (see page 370).
 - API 5C7 collapse model (see page 372).
 - Plastic collapse model (see page 372).
 - Safety factors (see page 372).
 - Imaginary data (see page 372).
 - Working limits.

4. Click **Calculate**.

 Hercules calculates the limits and displays the graph.

The Differential Pressures Curve

The Differential Pressures Curve is a conventional graph. It displays limits based on the difference between inner and outer pressures. (See "How the Differential Pressures Curve is Calculated" on page 371.)

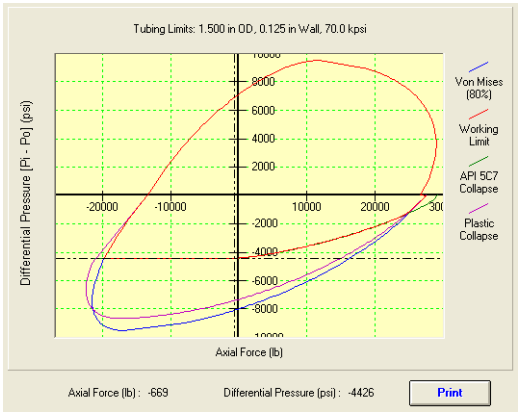
Reading the Differential Pressures Graph

The Differential Pressures Graph combines the internal and external pressure as a pressure difference. The x-axis is the axial load (how much tension or compression). The y-axis is the difference in pressure (inside minus outside pressure).

The ellipses on the graph represent the limits. Inside the red ellipse is the working area. The risk for tubing failure increases if a pressure and axial load combination outside the ellipse is used.

To...	Do this...
Find the exact coordinates of the cross-hairs.	Look at the bottom of the graph.
Zoom in on an area of the graph.	Click and drag a rectangle around the area to zoom in on.
Zoom out to see the entire graph.	Right-click the graph
Snap to Working Limit.	Select **Snap to Working Limit** under the Options tab.
Print the graph.	Click **File** and select **Print**.

Differential Pressures Graph



Troubleshooting in Hercules

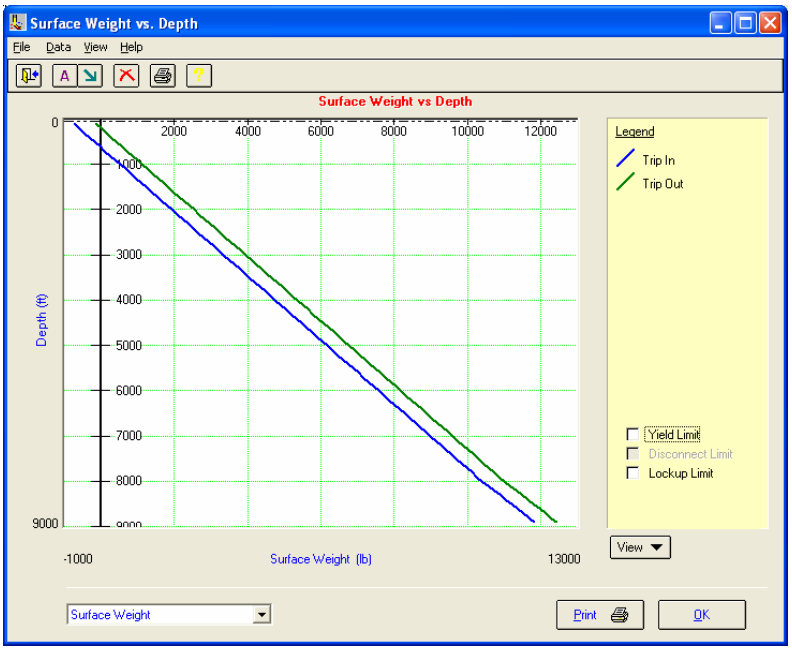
Situation	Possible Cause	Action
The left side of the graph is truncated.	Helical buckling has been included in the graph. The truncated shape is the result of the extra bending stress caused by helical buckling when the CT is in compression.	To turn off helical buckling, clear the check box on the main screen under the **Options** tab, and recalculate the graph.

Appendix A
Setting Graph Display Options in Orpheus

This section deals with the various options available for configuring the display of Surface Weight vs. Depth graphs. The user should note that the graph display-options described here generally pertain to all graphs generated as a result of Orpheus standard calculations.

Configuring Standard Calculation Graphs

As described in the Using Orpheus chapter of the *Cerberus User Guide*, a Surface Weight vs. Depth graph appears when the user selects, from the Calculate tab within an Orpheus project display screen, the **Trip In and Out** button. The initiated calculation plots a graph showing weight vs. depth as the CT/WL is tripped/run into the well (shown in blue) and out of the well (shown in green). The calculation is performed from the surface to the measured depth.



From the graph, the user may calculate Yield Limit, Disconnect Limit, and/or Lockup Limit by clicking (to toggle) the associated checkbox in the lower portion of the Legend panel found on the right-hand side of the display screen.

Calculating Yield Limit and Lockup Limit

The Yield Limit and Lockup Limit calculation procedures described here may be used as “templates” for other Orpheus standard calculations. See Standard Calculations for additional information.

➢ To calculate yield limit

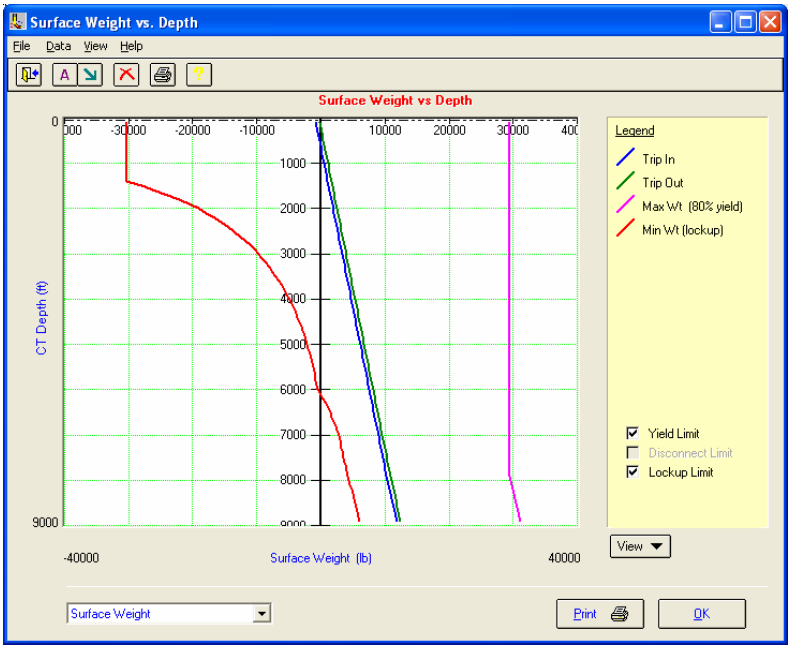
- Click the **Yield Limit** checkbox.

 The Max Wt (% yield) is displayed on the graph and within the legend.

➢ To calculate lockup limit

- Click the **Lockup Limit** checkbox.

The Min Wt (lockup) is displayed on the graph and within the legend.



Changing Default Curve-style Settings

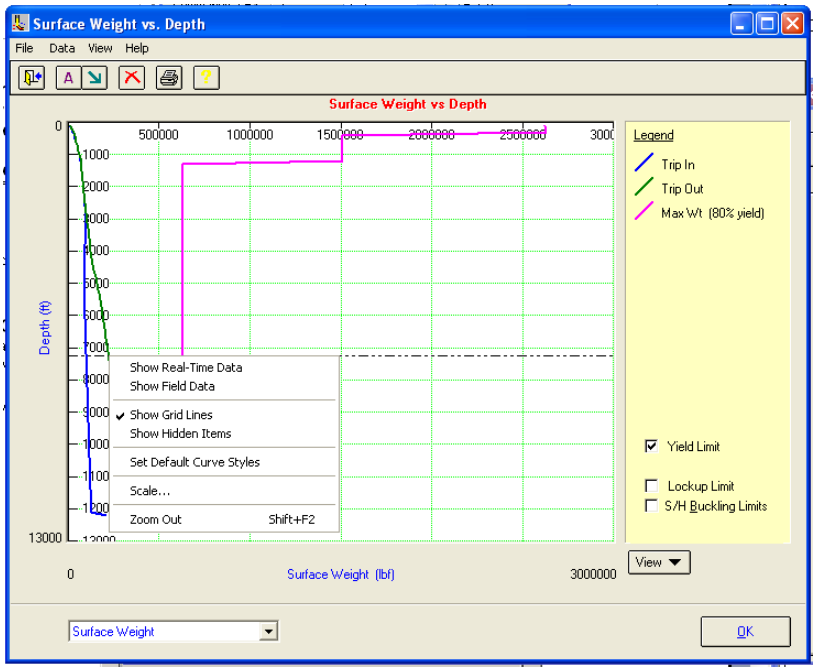
➢ To change default curve-style settings

1. Click (to select) a line in the graph or legend.

 The line begins to blink.

 a. Right-click the line (graph or legend).

 A menu list becomes visible.

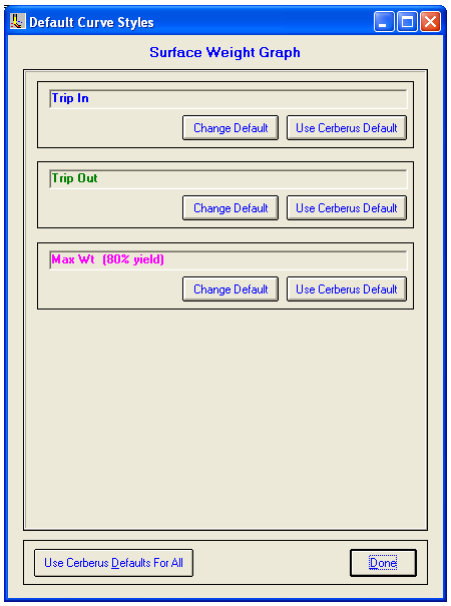


b. Select **Set Default Curve Styles**.

OR

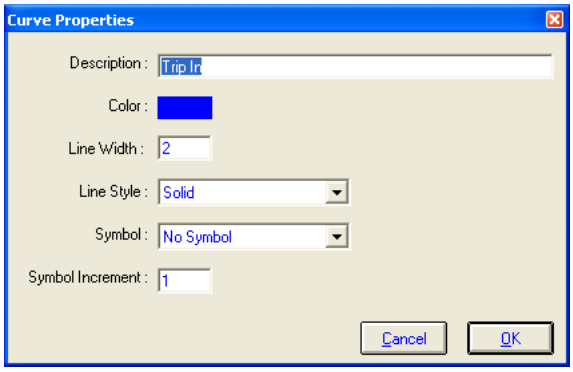
1. Select, from the menu bar of a graph, **View > Set Default Curve Styles**.

 The Default Curve Styles dialog appears.



The Default Curve Styles dialog shows curves belonging to the graph type displayed. Listed curves include those currently visible and those previously viewed. These settings are not applied to the "Automatic Report" in Report Generator.

2. Click the **Change Default** button related to any curve listed to alter that curve's properties via the Curve Properties dialog.



3. After making desired changes, click **OK** to save the altered curve style as the curve's new default style. These settings will be used any time this graph is recalculated.

4. To restore a curve's original Cerberus setting, click the **Use Cerberus Default** button related to the altered curve.

Changing Curve Properties

Especially useful when printing with no color (as in grayscale printing), the user may change the appearance of a line on the graph and/or legend to enhance differentiation between line types.

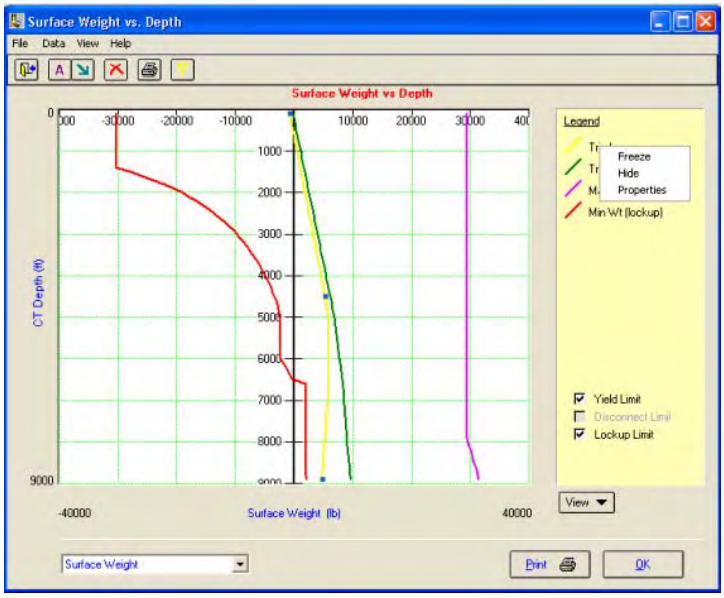
➢ To change curve properties

1. Click (to select) a line in the graph or legend.

 The line begins to blink.

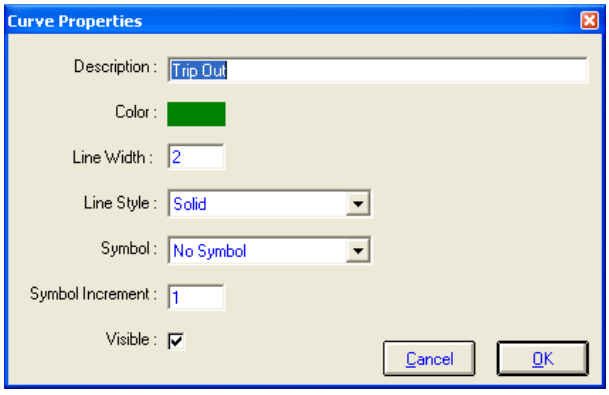
2. Right-click the line (graph or legend).

 A menu list becomes visible.



3. From the menu list, select **Properties**.

 The Curve Properties dialog appears.



4. Change the legend's description and line properties as needed.

➢ To change the line color

1. Click the **Color** box.
2. Select from available colors or define a custom color.

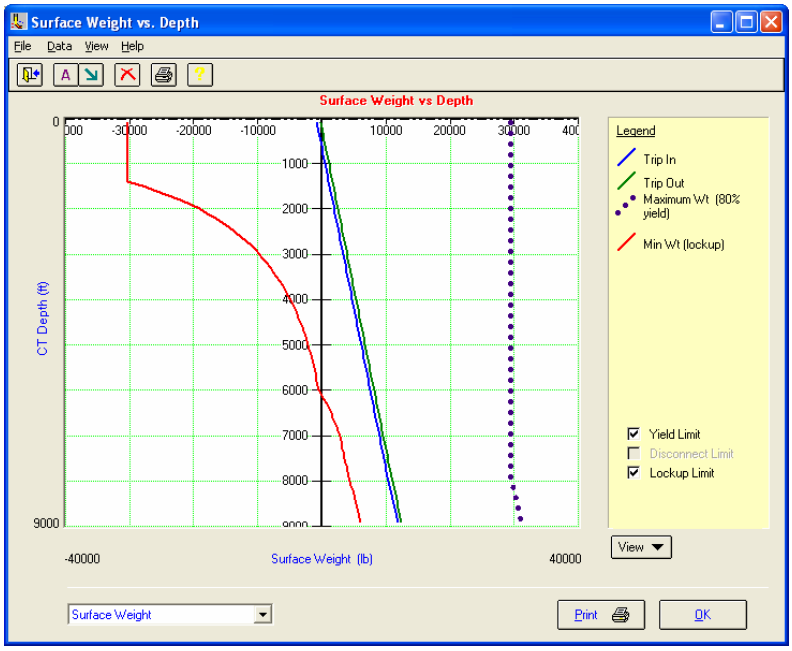
➢ To change the symbol

1. Select the **Symbol** from the dropdown.
2. Input the **Symbol Increment**.

NOV CTES recommends an increment of "5."

5. Click **OK** to save changes.

 The legend and line properties reflect the properties selected.



Freezing a Line on a Graph

After calculating Trip in and Out, freeze a line on the graph so that it displays as configured when the software is closed and re-accessed.

To retain line configurations, ensure that the project is saved prior to closing the software.

➢ To freeze a graph line

1. Click (to select) a line in the graph or legend.

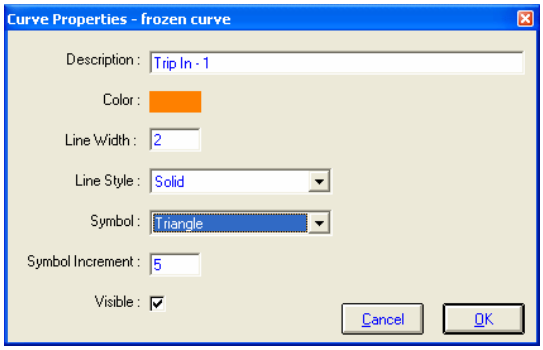
 The line begins to blink.

2. Right-click the line (graph or legend).

 A menu list becomes visible.

3. Select **Freeze** from the menu list.

4. The Curve Properties – Frozen Curve dialog appears.



In this example, the color, symbol, and symbol increment have been changed.

Note the automatic change of the curve-properties description.

5. Click **OK**.

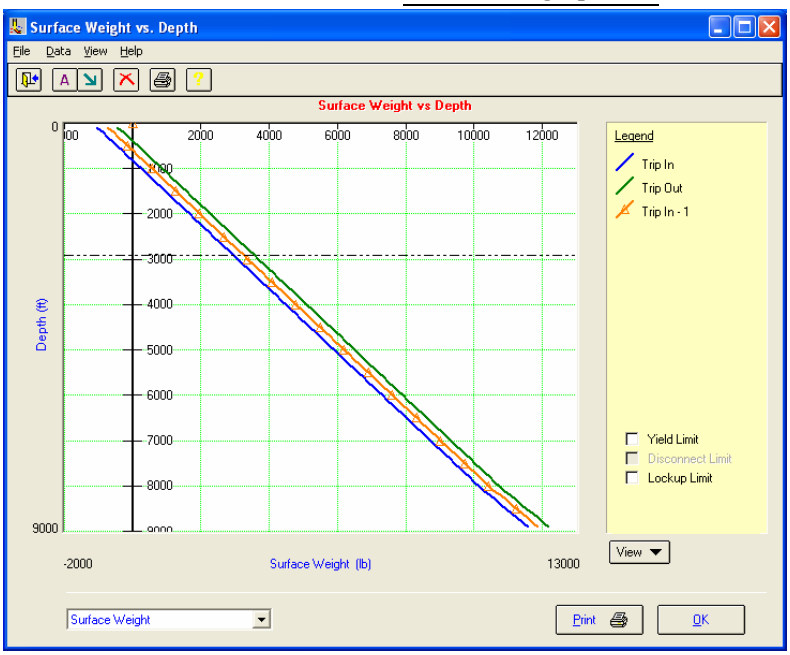
➢ To observe the frozen line in an altered project

1. Close the graph in which the line was frozen.

2. Alter the Surface Conditions and/or Downhole Conditions within the Calculate tab of the original Orpheus project.

3. Select the **Trip In and Out** button to re-run the calculation.

4. Observe the frozen line created in To freeze a graph line within the newly calculated graph.



Hiding a Line on a Graph

➢ To hide a graph line

1. Click (to select) a line in the graph or legend.

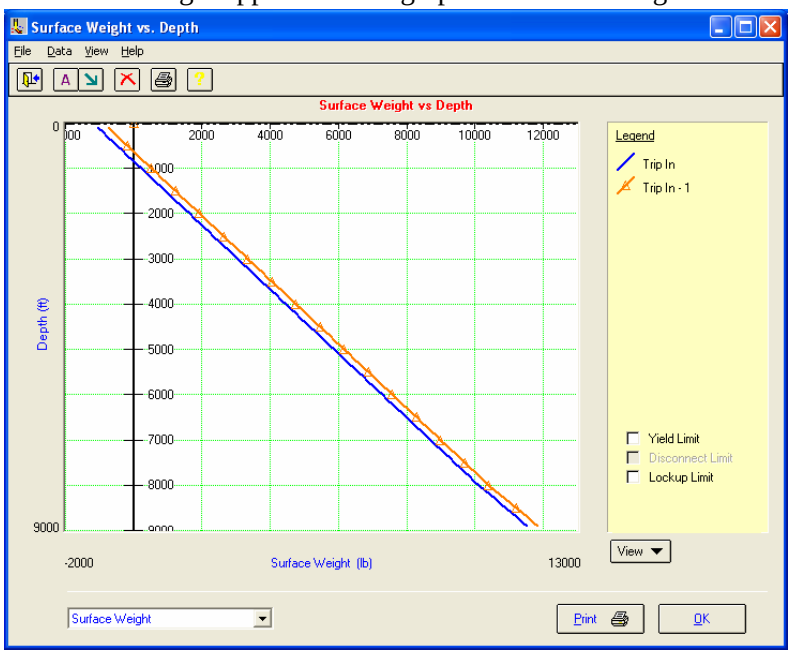
 The line begins to blink.

2. Right-click the line (graph or legend).

 A menu list becomes visible.

3. Select **Hide** from the menu list.

 The line no longer appears on the graph or within the legend.



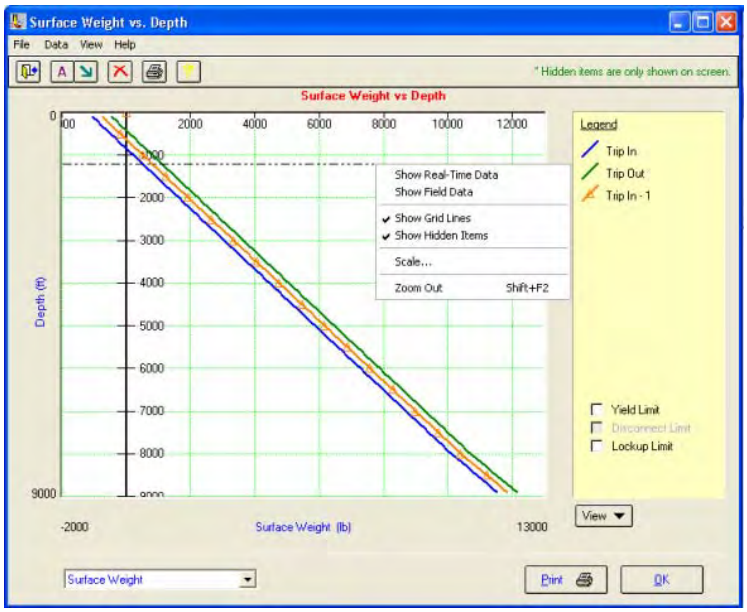
Showing Hidden Lines on a Graph

➢ To show all hidden lines on a graph

1. Right-click on the graph.

 The applicable menu list appears.

2. Select **Show Hidden Items** from the menu list.



All previously hidden lines are displayed on the graph and within the legend.

Appendix B
Coiled Tubing Forces Theory

This section deals with tubing forces theory. The links point to paragraphs that cover each of the following topics in greater detail.

- "Sinusoidal Buckling" on page 390.
- "Helical Buckling" on page 390.
- "Lockup Theory" on page 390.
- "Effect of Curvature on Helical Buckling Load" on page 391.
- "Residual Bend and Friction Coefficients" on page 391.
- "Real Force vs. Effective Force" on page 391.
- "Force vs. Weight" on page 392.
- "Factors Affecting Weight" on page 392.
- "Capstan or Belt Effect" on page 393.
- "Free Fall" on page 393.
- "How to Get More Set Down Force" on page 394.
- "Torque" on page 394.
- "Finite Element Analysis" on page 395.

Sinusoidal Buckling

Imagine a straight CT string being pushed into a straight horizontal casing. For the initial distance, the CT remains straight, lying nicely in the "trough" formed by the bottom of the casing. As the length of CT pushed into the casing increases, the force required to push it increases. This force is equal to the total weight of the CT string in the casing multiplied by the friction coefficient. As the length increases, the frictional drag increases, and thus the force required to push it increases.

When the force required to push the CT reaches a certain amount (load), the CT begins to "snake" in a sinusoidal fashion back and forth across the bottom of the casing. This load is referred to as the "**sinusoidal buckling load**" or sometimes the "snake buckling load." In drill pipe Tubing Forces Models (TFMs), this is often referred to as the "critical buckling load." However, there is nothing "critical" about this mode of buckling. The period of the sine wave is very large (usually 30 to 100 ft), and, of course, its amplitude is no greater than the internal diameter of the casing. Thus, the bending that is occurring is trivial. Sinusoidal buckling has no impact on the tubing forces calculation.

Helical Buckling

Imagine a straight CT string being pushed into a straight horizontal casing. The CT may form three portions. The first portion of the CT will still be lying straight in the casing. The second portion, which has an axial load greater than the sinusoidal bucking load, will lie in a sine wave in the bottom of the casing. In the third portion, the CT forms a helix inside the casing. The CT begins to form this helix at the "**helical buckling load** (HBL)."

The period of the helix at the onset of helical buckling is large, and no significant bending stresses occur in the CT material. However, at this point, the tubing forces calculation changes. As the helix is pushed into the casing, there are additional wall contact forces due to the helix. These wall contact forces increase the friction with the wall of the casing, causing less and less force to be transmitted downhole.



In a vertical well, the sinusoidal and helical buckling loads are small. Thus, buckling occurs with a small amount of compression.

The helical and sinusoidal buckling loads may be calculated based on either theoretical “loading” or “unloading” values. The loading values occur when the compressive load is increasing, and the pipe is going from straight to sinusoidal to helical. The unloading values occur when the pipe begins already helically buckled, and the compressive load is decreased until the pipe becomes sinusoidally buckled and then straight. The unloading values for the SSBL and HBL are smaller than the loading values.



NOV CTES recommends using the unloading values because the smaller buckling loads are more conservative (additional wall contact forces occur at smaller compressive loads).

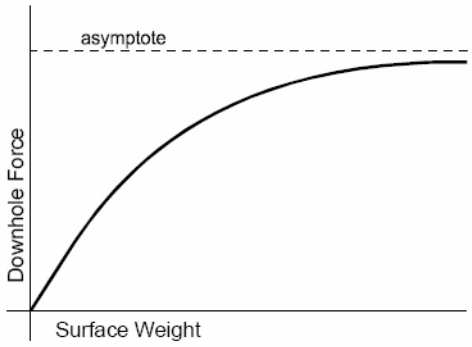
Lockup Theory

Imagine a straight CT string being pushed into a straight horizontal casing. The CT may form three portions. First, there is a straight portion up to the point where the sinusoidal buckling load is reached. This is followed by a portion which is buckled into a sine wave, until the helical buckling load is reached. Finally, there is a portion of the CT which is buckled into a helix. It is only in this third, helical portion for which additional wall contact forces are being generated.

If more force is applied at the surface to helically buckled CT, the wall contact forces increase faster than the transmitted force, and lockup eventually occurs. Lockup is a "vicious circle": pushing on the CT increases the helical buckling, which increases the wall contact forces, making it harder to push the CT. No matter how hard

the CT is pushed, no more than a certain amount of downhole force is achievable. Any additional CT pushed into the casing only increases the helical bucking.

The following graph shows the general relationship between the downhole force and the set down weight at a specific depth. The CT stays at a particular depth while more and more weight is applied at the surface, but less and less force reaches the end of the tool string.



The curve approaches a horizontal asymptote. Since the downhole force never reaches the asymptote, it is difficult to determine exactly when the CT is locked up. Orpheus has two different lockup definitions it can use to determine when the CT is locked up.

Effect of Curvature on Helical Buckling Load

Consider CT lying in a curved casing. The axial load applied to the CT causes it to "seat" itself in the "trough" formed by the casing. As the axial load increases, the radial load pushing the CT into the seat increases. Thus, the axial load required to cause the CT to pop out of the seat and form a helix is much greater than the helical buckling load for a straight hole.

Increasing the helical buckling load delays the onset of helical buckling, and thus delays the onset of lockup. Thus it could be argued that curvature in the well is beneficial. However, curvature in the well also causes the belt effect, which increases friction.

Residual Bend and Friction Coefficients

Bending that occurs to the CT at the reel and at the guide arch causes residual stresses in the CT material.

It was previously thought that the difference in the two friction coefficients (0.3 RIH, 0.25 POOH) in the well was due to residual bend. However, some testing with straight pipe and pipe with residual bend performed by CTES for Mobil showed that the two had the same friction. Other companies have done testing and claimed to have different results. The effect of residual bend is not fully understood.

Even though the reason for the difference in the friction coefficients is not understood, using a friction coefficient of 0.3 for RIH and 0.25 POOH has still proven to be accurate in many wells.

Real Force vs. Effective Force

Real force is the actual force on the pipe. It can be measured with a force measuring device, such as a strain gauge. The true stress and stretch of the pipe results from the real force.

Effective force is the force on the pipe accounting for buoyant weight and drag, but ignoring changes in internal and external pressure. Buckling and lockup are determined only by buoyant weight and are not affected by changes in internal or external pressure. Because real force changes with internal and external pressure, it cannot used for modeling buckling and lockup. Instead, effective force must be used.

The relationship between real force and effective force is shown in the following equation.

$$F_R = F_E + P_i A_i - P_o A_o$$

Orpheus uses effective force, instead of real force, in its downhole calculations. The effective force is then converted into real force only for stress and stretch calculations and for output purposes.

Force vs. Weight

The real force above the stripper is related to, but not equal to, the output of the weight sensor on the injector.

Real force above the stripper includes the effects of pressure, but omits reel back tension (RBT). However, the output of the weight sensor omits the effects of pressure, but includes reel back tension.

$$Weight = F_{Rabove\,stripper} - P_i A_i - RBT$$

Since effective force, by definition, does not include the effects of pressure (see “Real Force vs. Effective Force” on page 391), weight can be expressed simply as a combination of effective force above the stripper and reel back tension. Orpheus uses this calculation to predict the weight.

$$Weight = F_{Eabove\,stripper} - RBT$$

Note that the force (real or effective) below the stripper does not include stripper friction.

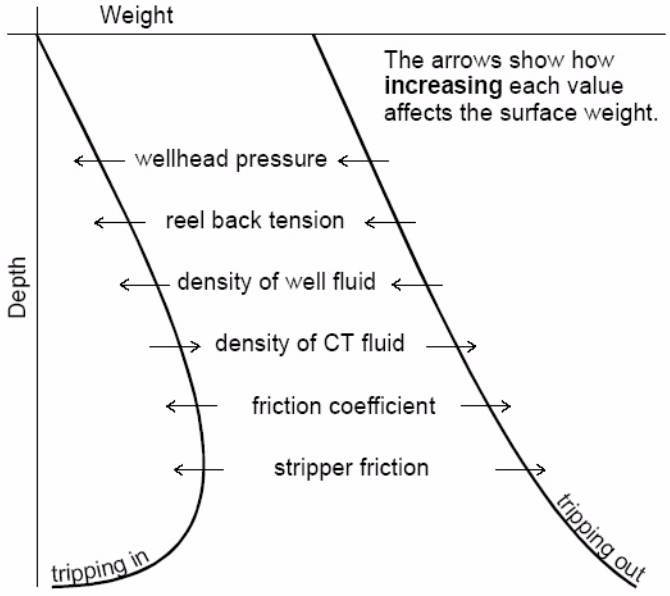
$$F_{above\,stripper} = F_{below\,stripper} \pm Stripper\,Friction$$

Factors Affecting Weight

The diagram below shows how increasing each of the following factors affects the surface weight reading.

The diagram below shows how increasing each of the following factors affects the surface weight reading.

The effects of the friction coefficients and the stripper friction depend on the direction the pipe is moving



Capstan or Belt Effect

Assume that a section of CT is in tension when it passes around a curve in a well. The tension causes the CT to be pulled against the inside of the curve. The greater the tension, the greater the radial load pushing the CT against the casing. This radial load causes the friction with the casing to increase. This increased friction is known as the "*capstan effect*" or "*belt effect*".

The same argument can be made if the CT is in compression. Now, the CT is pushed against the outside of the curve in the well. Again, additional friction forces are generated which must be considered in a tubing forces calculation.

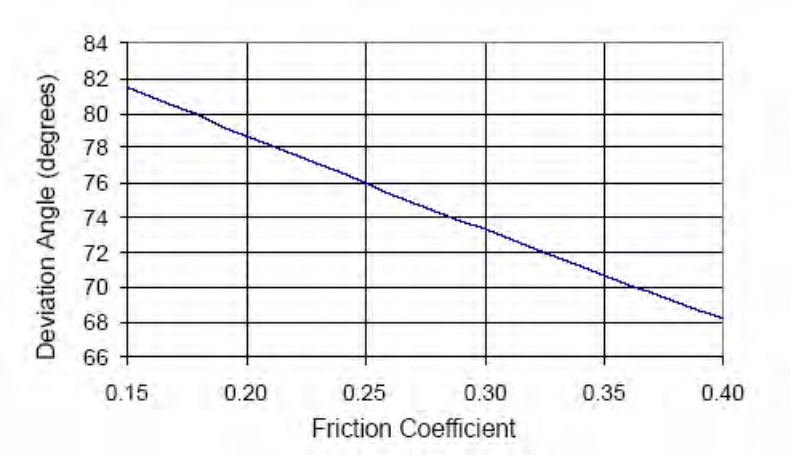
Thus, any curvature in a well, either in the inclination or the azimuth directions, causes additional friction which adversely affects the movement of the CT into and out of a well.

Free Fall

An object (such as CT or a cable with a tool on the end) can fall freely to the end of a vertical well without being pushed. As the deviation of a well increases, the falling object experiences more and more friction. If the deviation of the well is great enough, friction prevents the object from falling into the well. The point at which an object no longer falls freely into a well is a function of the friction coefficient of a well (μ) and its deviation angle (in radians – θ)

$$\theta = \arctan\left(\frac{1}{\mu}\right)$$

This equation yields the following plot. For points below the line, CT falls into the well. For points above the line, CT must be pushed or pulled to reach further.



For a typical friction coefficient of 0.3 running in hole, CT will fall freely into a well with a deviation angle less than 73 degrees.

How to Get More Set Down Force

The job type determines the amount of set down force needed. Some additional force should be available as an extra safety margin.

Some of the ways to get more set down force on the same string include:

- Increase the buoyancy of the string by pumping it full of nitrogen.
- Reduce friction by using lubricants or rollers on the tool string.
- Use a downhole tractor.
- Decrease the radial clearance by hanging a temporary liner or tubing string.

Some of the ways to get more set down force on a different string include:

- Use a CT string with a thicker wall.
- Use a CT string with a larger outside diameter.

Torque

Torque is a rotational force applied to the CT. It is typically created by a downhole motor. As torque travels up the length of the CT, it decreases in magnitude due to friction from contact with the wellbore wall. If the downhole torque is small enough and the wall contact force is great enough, the torque will not reach the surface.

In a vertical well, there is no wall contact force (unless the CT is buckled) and the full amount of torque is transmitted up the CT.

To calculate the torque along the length of a string in Orpheus, enter the torque on end created by the drilling motor, perform the run at measured depth calculation, and look at the torque graph.

Finite Element Analysis

For the most part, Orpheus uses what is known as a “soft-string” model. This type of model omits the bending stiffness of the pipe. When a pipe is bent around a significant curve or dog leg, there are additional wall contact forces and, thus, there is additional friction due to the bending stiffness of the pipe. In most wellbore applications, the additional friction due to this bending is small and can be ignored. Cases in which this bending stiffness becomes significant include very severe bends (such as elbows in a pipeline) and cases in which the pipe OD is close to the hole ID (such as running casing).

Orpheus uses a “stiff-string” Finite Element Analysis (FEA) model for the tool section of the string when using Tool Model 2.0. This model does take the bending stiffness into consideration. Often the tools, such as perforating guns, are significantly stiffer than the pipe which conveys them. They may also contain centralizers which could increase the wall contact forces.

Appendix C
Advanced Fluids-model Options in Orpheus

The modeling/calculating of hypothetical fluid-hydraulics situations in Orpheus may be accomplished using the techniques described here.

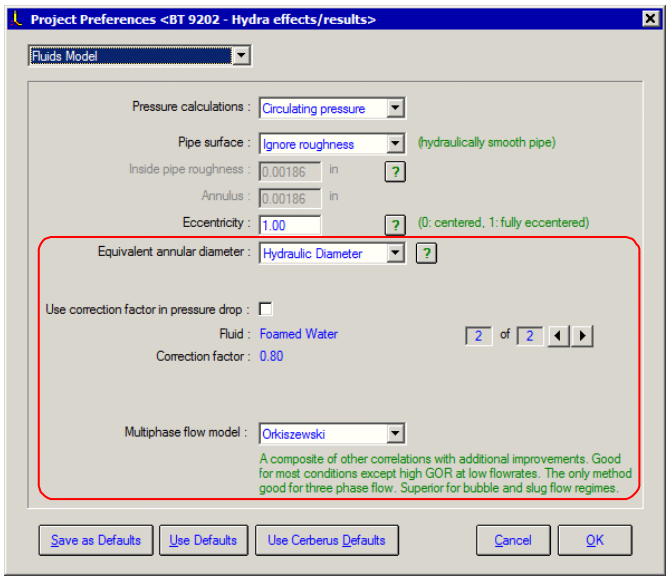
Fluids Model: Advanced Hydraulics Options

Hypothetical fluid-hydraulics situations may be modeled/calculated in Orpheus to determine the effects of differing variables.

As stated in the Entering Project Options section of this user guide's Using Orpheus chapter, the options set in the Advanced [project] Options are reflected in the calculation-parameter settings.

The options discussed in this appendix are available only with a Hydraulics-enabled access key. Contact Us for additional information.

After clicking the Advanced Options button and progressing to the Project Preferences display screen (see To set advanced options for additional information), select **Fluids Model** from the top-of-screen dropdown. A display screen similar to the following appears:



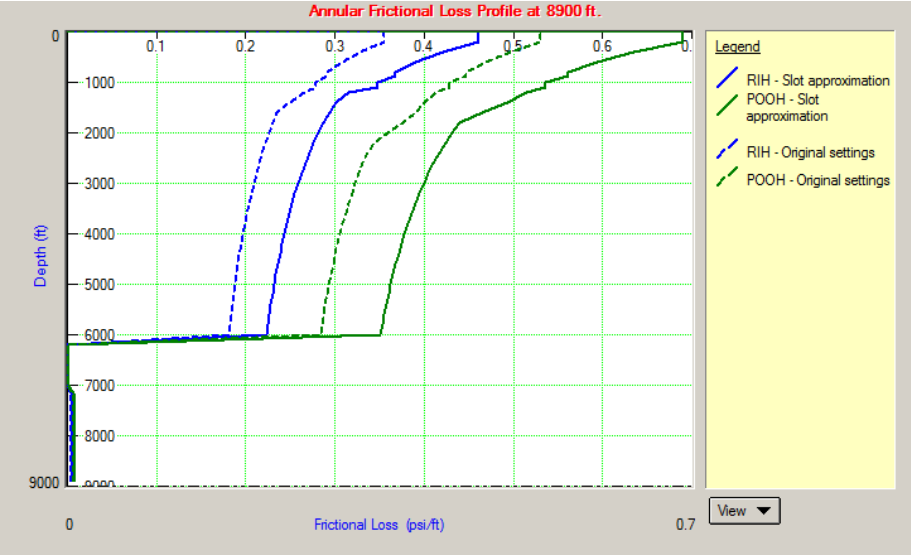
Within the red-box-highlighted portion of the above display screen, only the field **Equivalent annual diameter** is consistently available regardless of the type of fluid used. Potentially available fields (dependant on fluid type) include:

- Equivalent annular diameter
- Temperature effects on viscosity (available for liquid fluid types only)
- Drag reduction for Power Law fluids (available for the fluid rheology of Power Law only)
- Multiphase flow model

Modeling Examples

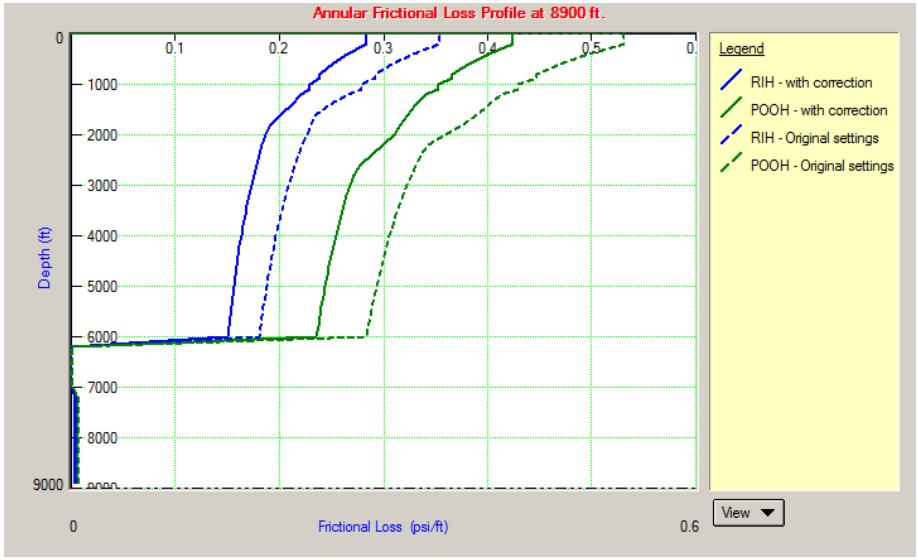
If the user simulates RIH and POOH calculations while freezing the resulting graph lines (see Freezing a Line on a Graph), the following types of information may be obtained:

Effect of Equivalent Annular Diameter



Effect of Correction Factor

See Fluid Correction Factors for additional information.



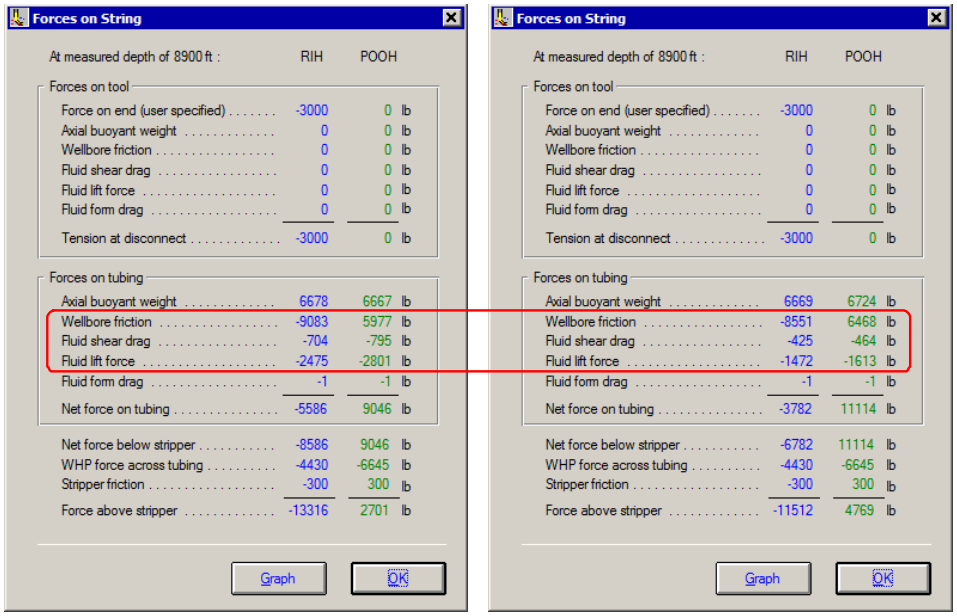
Effect of Temperature

To see the effect of temperature on viscosity, which is available for liquids only, change the fluid in the annulus to one for which temperature effects are defined.



Effect of Drag Reduction for Power Law Fluid

To see the effect of drag reduction for Power Law fluid, change the fluid in the annulus to a Power Law fluid type.



Effect of Multiphase Model

To model a multi-phase flow, select a multi-phase fluid type. The multiphase model controls which algorithm is used to predict the flow of multiphase fluids. It affects the flow regime, liquid holdup, velocities, density, and pressure drop of these fluids.

Appendix D
Features and Effects Options in Orpheus

As of publication date, this appendix describes the Orpheus Features and Effects settings related to:

- Modeling the Effects of the NOV Downhole Agitator Tool (CT/JP)
- Completing Jar Activation Calculations (WL)
- Including Additional Normal Force in Calculations (WL)

Modeling the Effects of the NOV Downhole Agitator Tool (CT/JP)

The Orpheus tubing forces model provides an analysis of the potential mitigation of string/wellbore friction made available by the Agitator tool.

The Agitator tool can mitigate a given amount of downhole friction (related to specific tool size, setup, and fluid pump rates). This friction mitigation effect provided by the tool gradually attenuates along the length of the CT string, and in normal situations a point is reached where there is no significant friction mitigation effect. From that point to surface, the friction is calculated and applied as it would be without the presence of an Agitator tool.

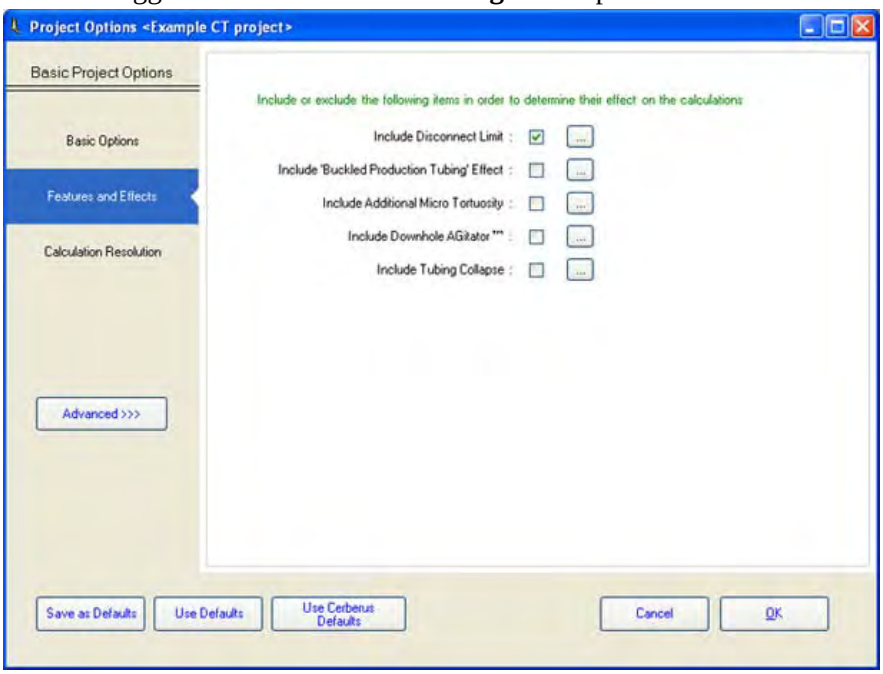
Including the Effect of the Agitator Tool in Calculations

➢ To include the effect of the Agitator Tool in calculations



1. Select **Options > Project Options** from the menu bar on the Orpheus project display screen; then select **Features and Effects** from the left-hand panel.

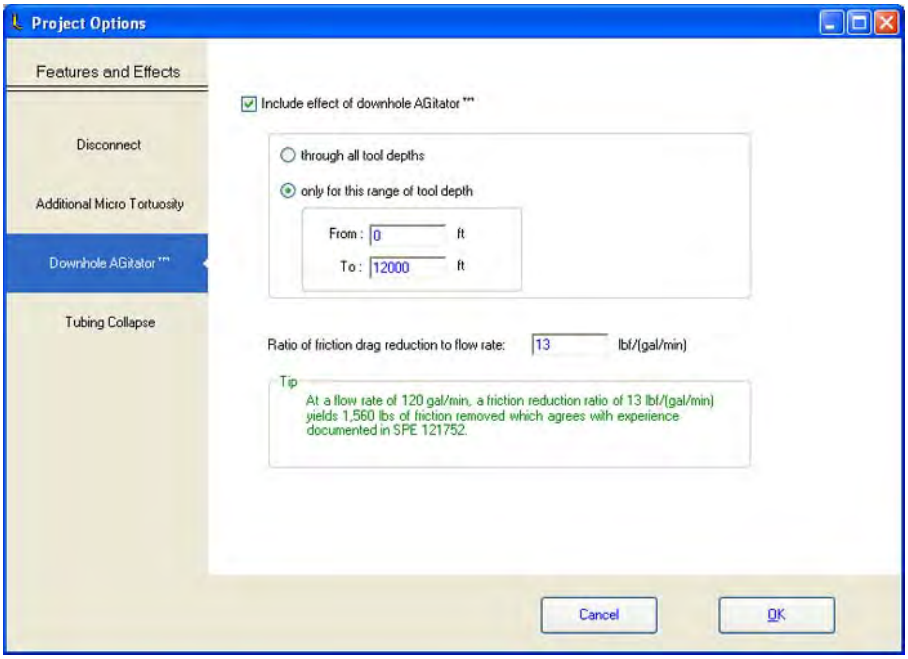
2. Click to toggle the **Include Downhole Agitator** option.



Enable the Agitator (to include its effect in downhole calculations) a few hundred feet above lockup depth for fastest calculation times. See Calculating Lockup Depth and Calculating Trip In and Trip Out for additional information concerning lockup depth.

3. To set the Agitator tool specifications, select the ellipses ([...]) associated with the **Include Downhole Agitator** option.

The [Project Options > Features and Effects] Downhole Agitator display screen appears.



4. Input the Agitator settings (a friction-drag-reduction to flow-rate ratio of 13 lbf/(gal/min) is recommended for 2-7/8" tools run in North America.[1] Agitator tools may be set up for more or less aggressive friction reduction in other regions of the world, and tools sizes other than 2-7/8" may have differing flow-rate ratios.

A flow rate of 120 gal/min is recommended for running the Agitator. The ratio of friction reduction multiplied by the flow rate (13 lbf/(gal/min) times 120 gal/min) results in the friction reduction of 1,560 lbs.[1]

5. Click **OK**. The software exits the Project Options display screen and returns the user to the main Orpheus project display screen.

Calculating Lockup Depth

➢ To calculate lockup depth

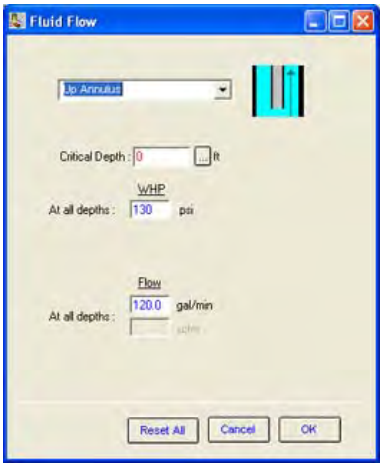
1. From the Orpheus project display screen, select the Calculate tab.

2. Underneath Downhole Conditions, input the **Liquid Flow Rate** as **120.0** gal/min.[1]

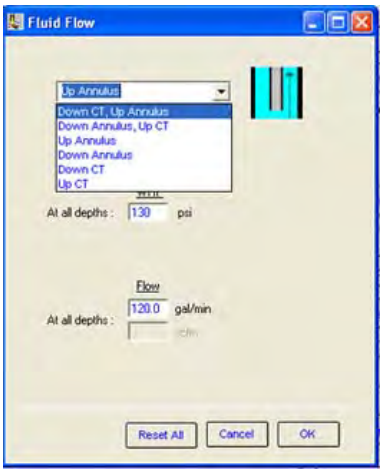
When calculating lockup depth or running Trip In and Out calculations, the flow rate should be set to the value used to run the Agitator (120 gal/min recommended)[1]*, even if the job does not require a high flow rate.*

1. Kenneth Newman and Timm Burnett, *Modeling the Affect of a Downhole Vibrator*, SPE 121752, 2009 SPE/ICoTA Coiled Tubing and Well Intervention Conference and Exhibition, Woodlands, Texas, USA, 31 March–1 April 2009.

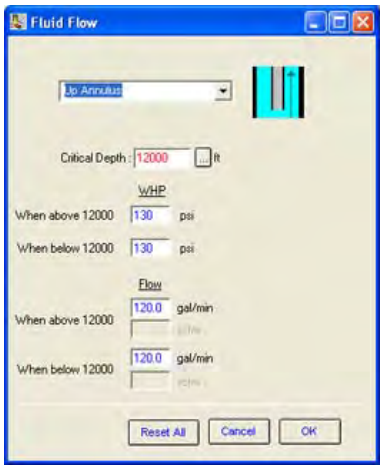
a. The user may select the associated with Liquid Flow Rate to reveal the Fluid Flow dialog.



Here, the user may select the fluid flow type from the associated dropdown, and select (after clicking the associated ellipses) **Critical Depth** as a function of **Max Survey Depth** or **Well TD**.

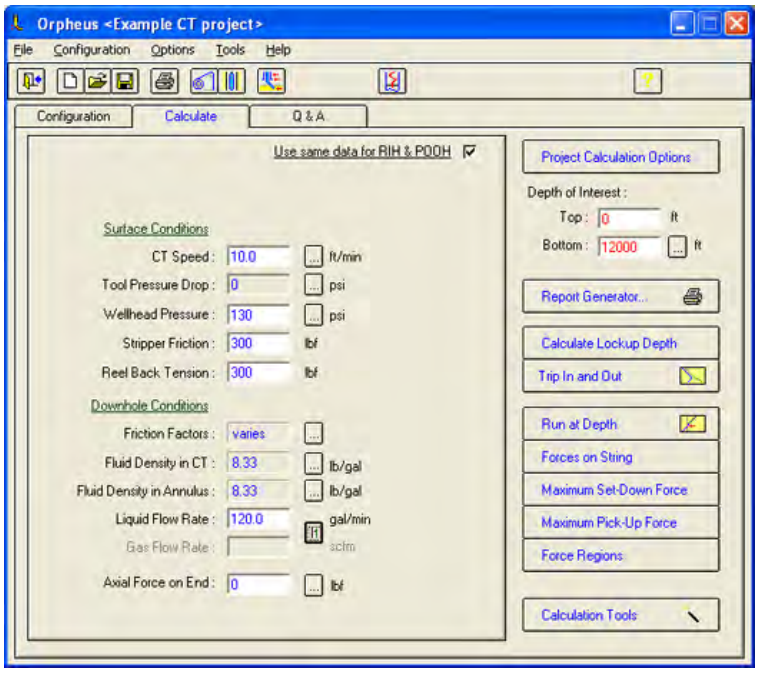


When **Critical Depth** function is selected (see directly above), the Fluid Flow dialog is altered to reflect wellhead pressure (**WHP**) and **Flow** above and below the **Critical Depth** designation. The user may alter these if desired.

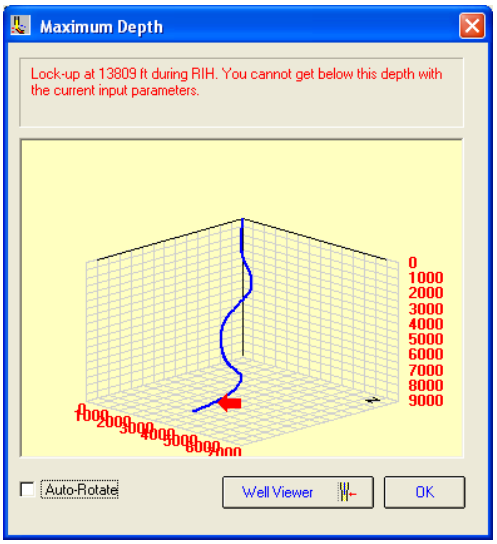


b. Click **OK** to accept the fluid-flow rates as entered, **Reset All** to return the dialog to the properties displayed prior to all changes, or **Cancel** to exit the dialog with no changes.

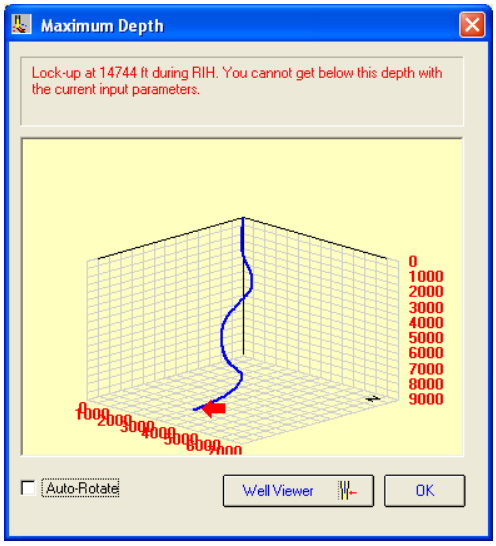
3. Click **Calculate Lockup Depth**.



The Maximum Depth dialog appears. In the case below, the Agitator effect is not enabled. Lockup occurs at 13,809 feet during RIH.



In the case below, the same downhole modeling scenario is used as above, but the Agitator effect is enabled. Lockup occurs at 14,744 feet during RIH. Thus the use of the Agitator tool under this set of downhole conditions delays the onset of lockup by 935 feet.



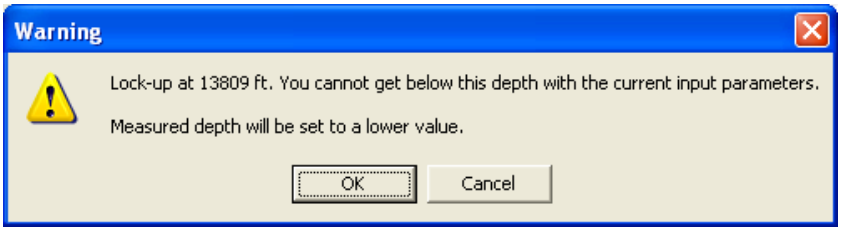
Calculating Trip In and Trip Out

➢ To calculate trip in and trip out

1. From the Orpheus project display screen, select the Calculate tab.
2. Underneath Downhole Conditions, input the **Liquid Flow Rate** as **120.0** gal/min.[1]
3. Click **Trip In and Out**.

 In the case below, the Agitator effect is not enabled.

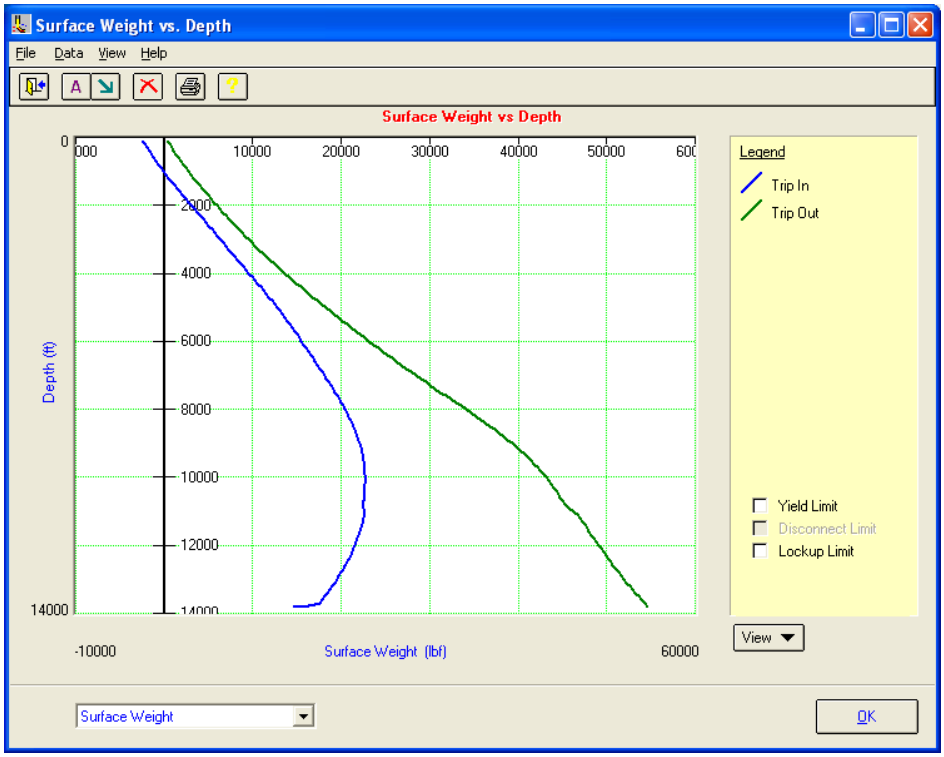
 a. A Warning dialog appears.



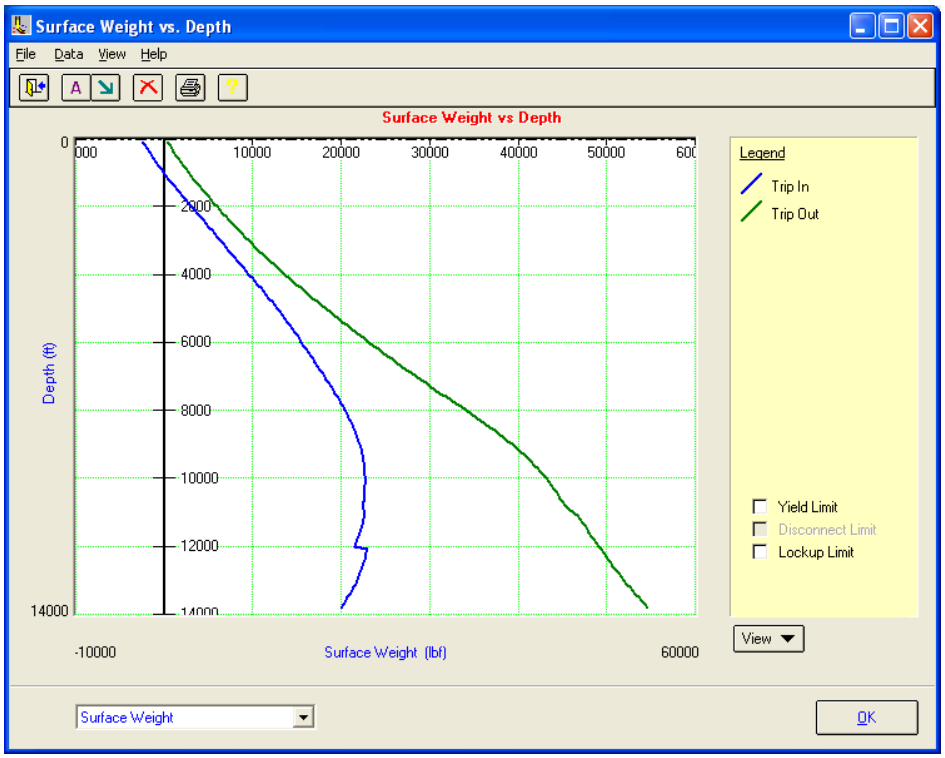
 The coiled tubing encounters lockup conditions at 13,809 feet when the Agitator is not enabled.

 b. Click **OK** to continue.

 The adjusted Surface Weight vs. Depth dialog appears.


Surface Weight vs. Depth
File Data View Help
Surface Weight vs Depth
0
10000
20000
30000
40000
50000
2000
4000
6000
8000
10000
12000
14000
Depth (ft)
-10000
Surface Weight (lbf)
60000
Legend
Trip In
Trip Out
Yield Limit
Disconnect Limit
Lockup Limit
View
Surface Weight
OK

In the case below, the Agitator effect is enabled for the depth range of 12,000 feet to 15,000 feet, so the calculation can proceed without further adjustment, and the coiled tubing is able to reach target depth.



Completing Jar Activation Calculations (WL)

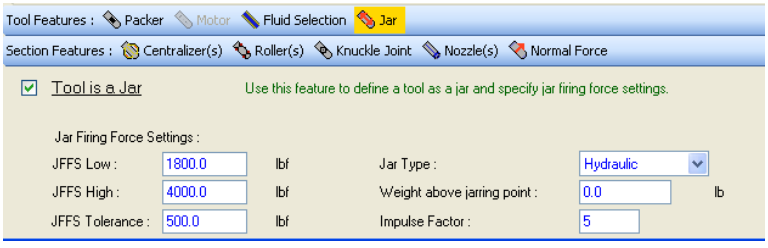
For Orpheus to perform a jar activation calculation, the tool string must contain a jar, and the option **Include Jar Activation Calculation** in the Orpheus Features and Effects section of the Options tab must be checked.

➢ To complete jar activation calculations in Orpheus

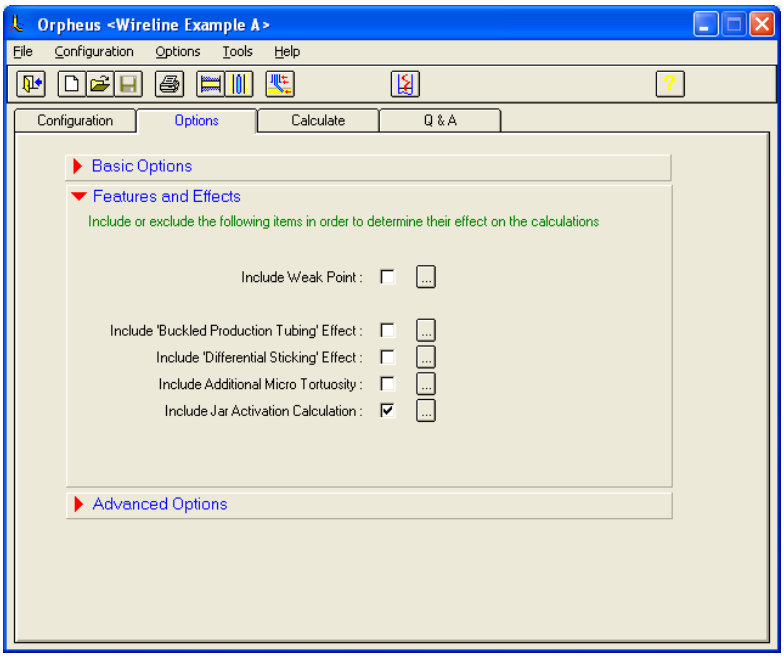
1. Access the Tool Editor's Tool Features functionality.

2. Set the tool to be involved in the jar activation calculation as a jar tool.

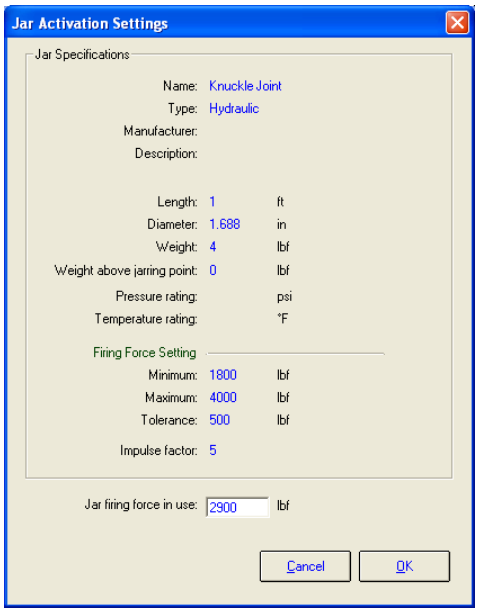
 a. Set the associated Jar Firing Force Settings (see Setting Jar Tool Features (WL)).



3. In Orpheus, from the Project Options, select the Features and Effects right-pointing arrow, and click to check **Include Jar Activation Calculation**.

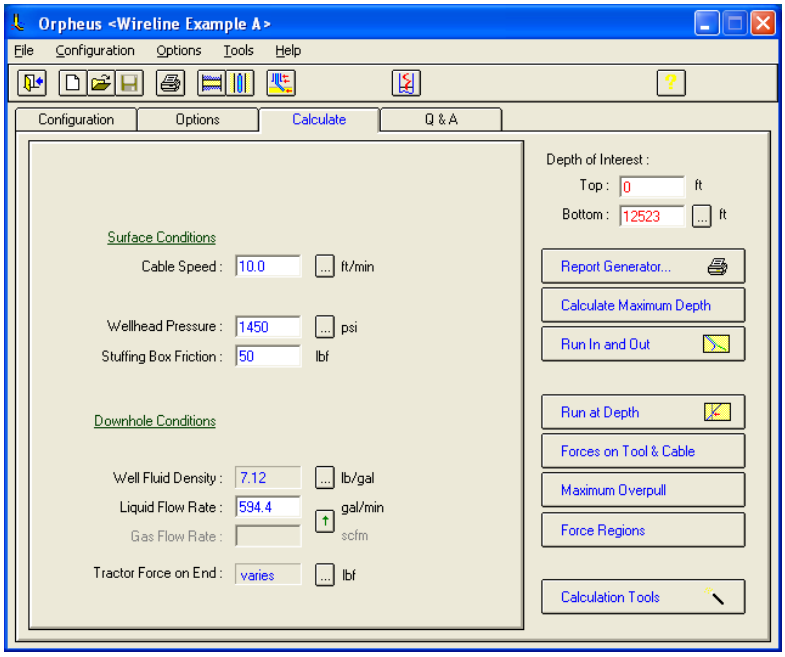


4. Click the associated ellipses button to access the Jar Activation Settings dialog.



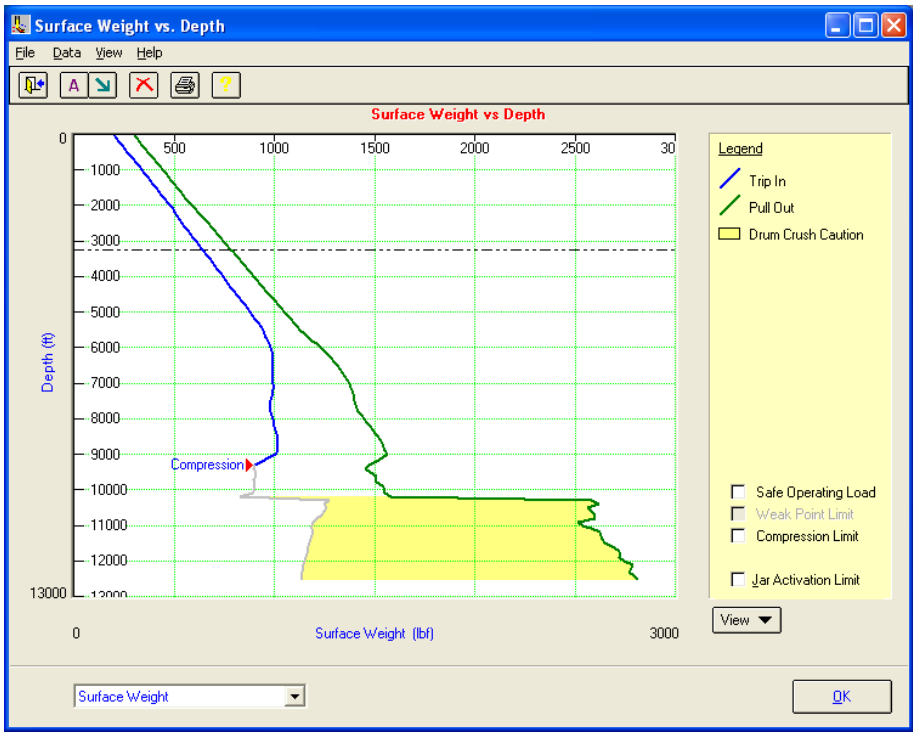
 a. In the **Jar firing force in use** field, set the jar firing force to a value between the **Minimum** and **Maximum Firing Force Setting** shown in the dialog, and click **OK**.

5. Select the Calculate tab, which is adjacent to the Options tab.

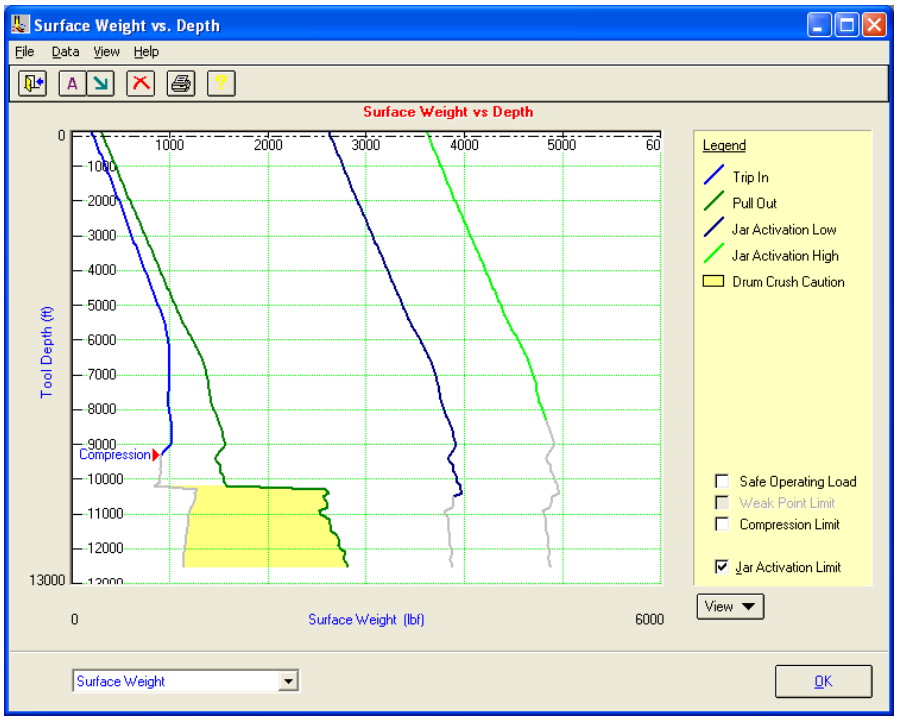


6. Select **Run In and Out**.

 Surface Weight vs. Depth is calculated.

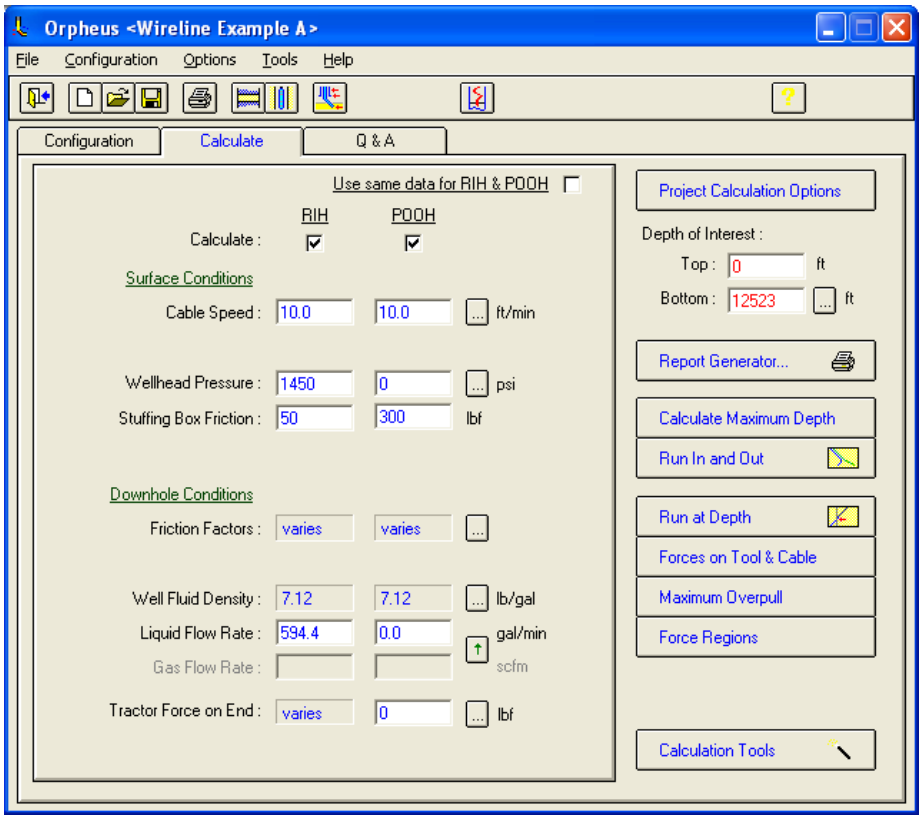


7. Click (to check) **Jar Activation Limit**, and click **OK**.



Including Additional Normal Force in Calculations (WL)

Application of additional normal force is specified in the Tool Editor (see Using Additional Normal Force in Calculations) and is applied in Orpheus.



Upon application of additional normal force, effective force increases as *Additional Normal Force * Well Friction*.

➢ To include additional normal force in calculations

1. If the same additional-normal-force data is desired for the RIH operation and the POOH operation, complete the following:

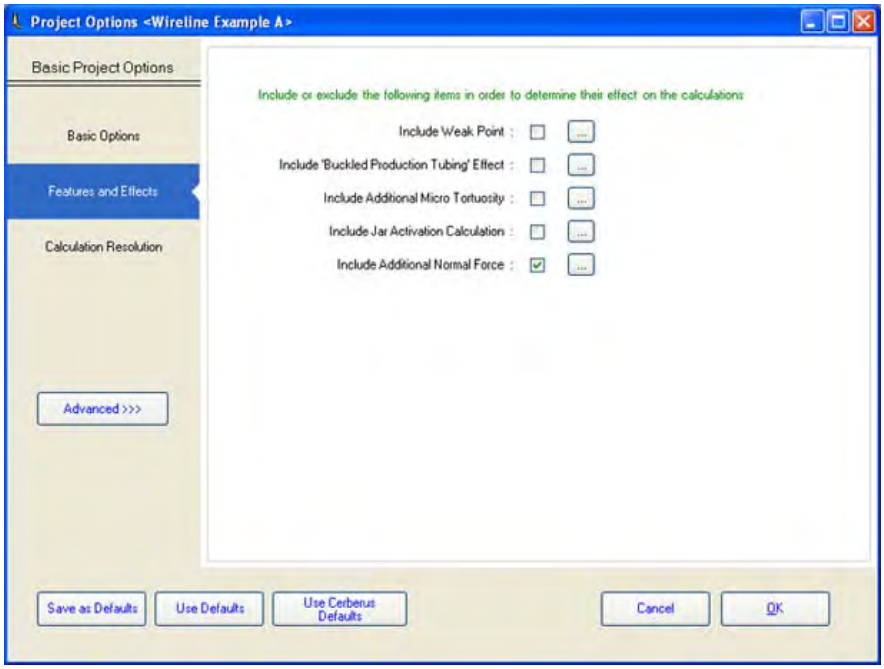
 a. Select the Calculate tab from the Orpheus project display screen.

 b. Click (to check) the box **Use same data for RIH & POOH**.

 The display screen changes from showing separate parameters for RIH and POOH to showing one set of parameters.

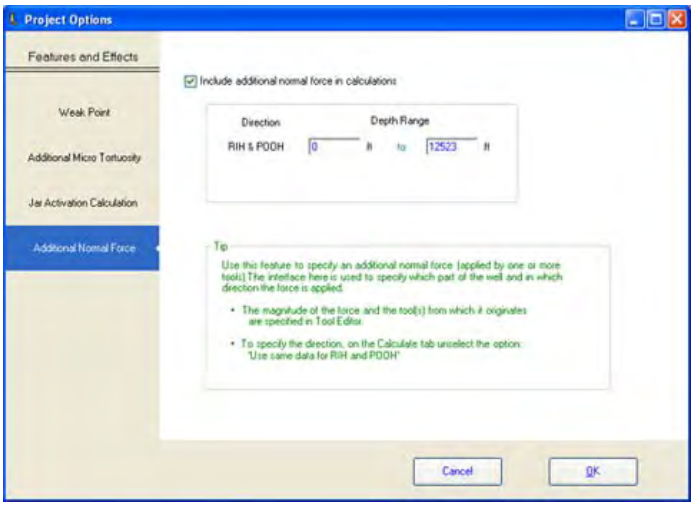
 c. Enter/update available parameters, as needed.

2. Select **Options > Project Options** from the menu bar on the Orpheus project display screen; then select **Features and Effects** from the left-hand panel. Click to check **Include Additional Normal Force**.

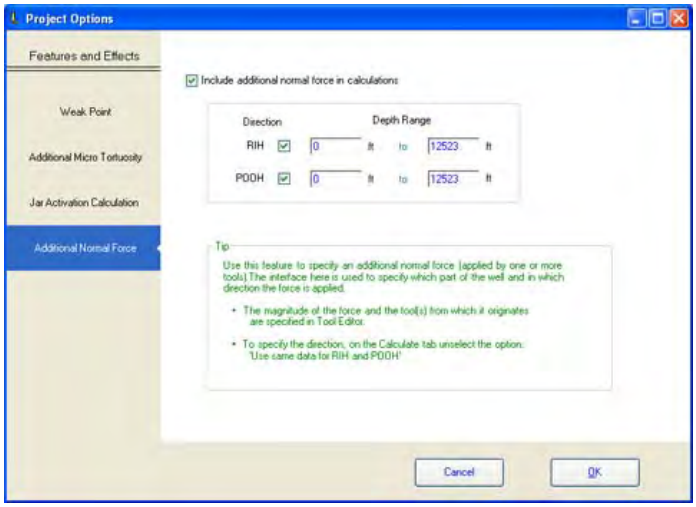


3. Click the associated ellipses button to specify **Depth Range**.

a. If the user chose to use the same data for RIH and POOH as described in Step 1, above, the Project Options display screen appears as shown below. Note that only one Direction/Depth Range is available for edit.



b. If the user chose to use different data for RIH and POOH as described in Step 1, above, the Project Options display screen appears as shown below. Note that separate Direction/Depth Range information is available for edit.



When using additional normal force, the resulting effective force becomes more positive when pulling out of the hole, and more negative when running into the hole.

Appendix E
Suggested Readings

- "Suggested Readings on Fatigue" on page 416.
- "Suggested Readings on Forces" on page 417.
- "Suggested Readings on Limits" on page 418.
- "Suggested Readings on Hydraulics" on page 418.

Suggested Readings on Fatigue

1. Brown, P., and Dickerson, J. L., “Development and Use of an Analytical Model to Predict Coiled Tubing Diameter Growth,” SPE 38409, 2nd SPE/ICoTA North American Coiled Tubing Roundtable, Montgomery, TX, April 1997.

2. Brown, P.A.: “Coiled Tubing Fatigue Modelling and Software Development”, PNEC Coiled Tubing Technology and Applications, February 6-8, 1995.

3. Brown, P.A.: “CT Life Modelling and Tracking”, World Oil 3rd International Conference and Exhibition on CT Technology, Houston TX, March 13-16, 1995.

4. Brown, P.A.: “Use of Fatigue Test Machine to Investigate Coiled Tubing Diameter Growth”, World Oil 2nd International Conference and Exhibition on Coiled Tubing, Amsterdam, June 1994.

5. Kane, R. and Cayard, M., "Factors Affecting CT Serviceability," *Coiled Tubing*.

6. Koper, M.G.M., Tan, S., and Rosen, P., "Full-Scale, Low-Cycles Fatigue Tests with 2-in Coiled Tubing and an Automatic Coiled Tuibng Inspection and Monitoring System," SPE 38415, 2nd North American Coiled Tubing Roundtable, Montgomery, TX April 1997.

7. Newman, K. and Allcorn, M., "CT in High Pressure Wells," SPE Annual Technical Conference, SPE 24793, October 1993.

8. Newman, K. and Newburn, D.: "Coiled Tubing Life Modelling," SPE paper 22820, 66th Annual Technical Conference and Exhibition, Dallas, October 1991.

9. Newman, K., Brown, P., Van Arnam, D., Wolhart, S.: ”Analysis of Coiled Tubing Welding Techniques,” SPE 36346, SPE/ICoTA 1st North American Coiled Tubing Roundtable, Montgomery, TX, February 1996.

10. Newman, K., Brown, P.: "Development of a Standard Coiled-Tubing Fatigue Test," SPE 26539, SPE Annual Technical Conference and Exhibition, Houston, October 1993.

11. Newman, K.: "Determining the Working Life of a Coiled Tubing String," *Offshore*, December 1991.

12. Quigley, M. S., Stone, L.C., “The Benefits of Real-Time Coiled Tubing Diameter Measurements” SPE 46040 3rd Annual SPE/ICoTA North American Coiled Tubing Roundtable, April 15-16, 1998.

13. Smith, Lawrence W., "Methods of Determining the Operational Life of Individual Strings of Coiled Tubing," Proceedings at the SPE Workovers and Well Intervention Seminar, November 16, 1989, Aberdeen, Scotland.

14. Tipton, S.M. & Brown, P.A.: “Monitoring Coiled Tubing Fatigue Life”, World Oil 2nd International Conference on Coiled Tubing Operations, Houston TX, March 28-31, 1994.

15. Tipton, Steven M. and Dale A. Newburn, "Plasticity and Fatigue Damage Modeling of Severely Loaded Tubing," 1st ASTM Symposium on Advances in Fatigue Lifetime Predictive Techniques, San Francisco, CA, April 1990.

16. Weld Cycle Life Study, Joint Industry Project DEA-97, 1995. Contact CTES for further information.

17. World Oil, *Coiled Tubing Handbook*.

Suggested Readings on Forces

1. McSpadden, A., and Newman, K., “Development of a Stiff String Forces Model for Coiled Tubing,” SPE 74831, 2002 SPE/ICoTA Coiled Tubing Roundtable, April 2002.

2. McSpadden, A., and Brown, P., “Field Validation of 3-Dimensional Drag Model for Tractor and Cable Conveyed Well Intervention,” SPE 71560, 2001 SPE Annual Technical Conference & Exhibition, 30 September - 3 October 2001.

3. Newman, K., and McSpadden, A., “Intervention in Wells with Buckled Production Tubing,” SPE 64501, 2000 SPE / ICoTA Coiled Tubing Conference and Exhibition, April 2000.

4. Newman, K., and Ackers, M., Stein, D., “Rotation of Coiled Tubing,” SPE 60737, 2000 SPE/ICoTA Coiled Tubing Conference and Exhibition, April 2000.

5. Newman, K., “Coiled Tubing Stretch and Stuck Point Calculations,” SPE 54458, SPE/ICoTA Coiled Tubing Conference and Exhibition, May 1999.

6. Adrichem, W. and Newman, K.: "Validation of Coiled-Tubing Penetration Predictions in Horizontal Wells," SPE paper 24765, 67th Annual Conference and Exhibition, Washington D.C., October 1992.

7. Gu, H., Newman, K., Hauglund, L.: "Analysis of Slack-off Force Transmitted Downhole in Coiled Tubing Operations," SPE paper 26511, SPE Annual Technical Conference and Exhibition, Houston, October 1993.

8. Newman, K., “Coiled Tubing Forces and Stresses Modelling Improvements”, 2nd ICoTA/SPE European CT Roundtable, October 1995.

9. Newman, K., Aasen, J.: “Catastrophic Buckling of Coiled Tubing in the Injector” 3rd North American Coiled Tubing Roundtable, Houston, TX. April 15-16, 1998.

10. Newman, K., Corrigan, M. and Cheatham, J.: "Safely Exceeding the Critical Buckling Load in Highly Deviated Holes," SPE paper 19229, Offshore Europe, Aberdeen, September 1989.

11. Newman, K., Sathuvali, U.B., and Wolhard, S., "Elongation of Coiled Tubing During its Life," SPE paper 38408, 2nd North American Coiled Tubing Roundtable, Montgomery, TX. April, 1997

12. Palmer, R., Newman, K., Reaper, A.: "Developments in Coiled Tubing BOP Ram Design," OTC Paper 7876, Offshore Technology Conference, Houston, May 1995.

13. Quigley, M. S., "Advanced Technology for Laboratory Measurements of Drilling Fluid Friction Coefficient," SPE 19537, SPE Annual Technical Conference, San Antonio, TX, October 8-11, 1989.

14. Quigley, M. S., A. K. Dzialowski, and M. Zamora: "A Full-Scale Wellbore Friction Simulator," IADC/SPE 19958, IADC/SPE Drilling Conference, Houston, TX, February 27-March 2, 1990.

15. Quigley, M. S., D. B. Lewis, and R. S. Boswell: "Brief: Field Measurements of Casing Tension Forces," SPE 30138, *Journal of Petroleum Technology*, February 1995, pp. 127-8.

16. Quigley, M. S., D. B. Lewis, and R. S. Boswell: "Field Measurements of Casing Tension Forces," SPE 28326, SPE Annual Technical Conference, New Orleans, LA, September 25-8, 1994.

Suggested Readings on Limits

1. McSpadden, A., Newman K., “Modified CT Limits Analysis for Practical Well Intervention Design,”SPE 74828, 2002 SPE/ICoTA Coiled Tubing Conference and Exhibition, April 2002.

2. Brown, P., and Dickerson, J. L., “Development and Use of an Analytical Model to Predict Coiled Tubing Diameter Growth,” SPE 38409, 2nd SPE/ICoTA North American Coiled Tubing Roundtable, Montgomery, TX, April 1997.

3. Craig, S. and Maroli, R., "Development of a 10,000 psi Working Presure Coiled Tubing Unit," 2nd International Conference & Exhibition on Coiled Tubing, Amsterdam, The Netherlands, June 1994.

4. Newman, K., Sathuvalli, U., Wolhart, S.: "Defining Coiled Tubing Limits - A New Approach,” OTC 8221, 28th Annual OTC, Houston, May 1996.

5. Newman, K.: "Coiled Tubing Pressure and Tension Limits," SPE paper 23131, Proceedings from Offshore Europe 91, Aberdeen, September 1991.

6. Newman, K.: "Collapse Pressure of Oval Coiled Tubing," SPE paper 24988, European Petroleum Conference, Cannes France, November 1992.

7. Timoshenko, S.P. and Gere, J.M. *Theory of Elastic Stability,* 2nd Edition, McGraw Hill, NY 1961.

8. van Adrichem, W.P., Gordon, D.G., and Newlands, D.J., "Development and Utilization of a Coiled Tubing Equipment Package for Work in High Pressure Wells," OTC paper 7874, Offshote Technology Conference, Houston, TX, May 1995.

Suggested Readings on Hydraulics

Multiphase Flow Readings

1. Rao, B., “Friction Factors for Turbulent Flow of Non-Newtonian Fluids in Coiled Tubing,” 2002 SPE / ICoTA Coiled Tubing Conference and Exhibition, April 9-10, 2002

2. Gu, H., Walter, I.C., Stein, D., “Designing Under- and Near Balanced Coiled Tubing Drilling by Use of Computer Simulations,” SPE 56864 SPEDC, June 1999.

3. Aziz, K., Govier, G. W., and Fogarasi, M.: “Pressure Drop in Wells Producing Oil and Gas,” *J. Canadian Pet. Tech.* (Jul.-Sept. 1972) 38-48.

4. Beggs, H. D. and Brill, J. P.: “A Study of Two-Phase Flow in Inclined Pipes,” *J. Pet. Tech.* (May 1973) 607-617.

5. Duns, H., Jr. and Ros, N. C. J.: “Vertical Flow of Gas and Liquid Mixtures in Wells,” *Proc. Sixth World Pet. Congress,* Frankfurt (Jun. 19-26, 1963) Section II, Paper 22-PD6.

6. Espanol, J. H., Holmes, C. S., and Brown, K. E.: "A Comparison of Existing Multiphase Flow Methods for the Calculation of Pressure Drop in Vertical Wells," SPE 2553 presented at the 44th Annual Fall Meeting of the Society of Petroleum Engineers of AIME, Denver, CO (Sept. 28-Oct. 1, 1969).

7. Griffith, P.: "Two-Phase Flow in Pipes," Special Summer Program, Massachusetts Institute of Technology, Cambridge, Massachusetts (1962).

8. Griffith, P. and Wallis, G.B.: "Two-Phase Slug Flow," *J. Heat Transfer; Trans. ASME* (Aug. 1961) 307-320.

9. Hagedorn, A. R. and Brown, K. E.: “Experimental Study of Pressure Gradients Occurring During Continuous Two-Phase Flow in Small Diameter Vertical Conduits,” *J. Pet. Tech.* (Apr. 1965) 475-484.

10. Lawson, J.D. and Brill, J.P.: "A Statistical Evaluation of Methods Used to Predict Pressure Losses for Multiphase flow in Vertical Oilwell Tubing," *J. Pet. Tech.* (Aug. 1974) 903-914.

11. Orkiszewski, J.: “Predicting Two-Phase Pressure Drops in Vertical Pipe,” *J. Pet. Tech.* (Jun. 1967) 829-838.

12. Sas-Jaworsky II, A.: “Coiled Tubing …Operations and Services, Part 5-Unloading Wells with Lighter Fluids” *World Oil* (Mar. 1992) 36-43.

13. Tek, M. R.: “Multiphase Flow of Water, Oil, and Natural Gas Through Vertical Flow Strings,” *J. Pet. Tech.* (Oct. 1961) 1029-1036.

14. Vohra, I.R., Robinson, J.R., Brill, J.P.: "Evaluation of Three New Methods for Predicting Pressure Losses in Vertical Oilwell Tubing," *J. Pet. Tech.* (Aug. 1974) 829-832.

Additional Readings

1. API Bulletin on Rheology and Hydraulics of Oil Well Drilling Fluids, Third Edition, (Jan. 1993).

2. Bourgoyne Jr., A. T., Chenevert, M. E., Millheim, K. K., Young Jr., F. S.: “Chap. 4: Drilling Hydraulics” SPE Textbook Series, Vol. 2, SPE Richardson, TX, (1986), 137-156.

3. Blauer, R. E., Mitchel, B. J., and Kohleas, C. A.: “Determination of Laminar, Turbulent, and Transitional Foam Flow Losses in Pipes” SPE 4885 presented at the 44th Annual California Regional Meeting of the Society of Petroleum Engineers of AIME, San Francisco, CA April 4-5, (1974).

4. Brown, P. T. and Wimberley, R. D.: “Coiled Tubing …Operations and Services, Part 10-Velocity Strings” *World Oil* (Jan. 1992) 77-80.

5. Sas-Jaworsky II, A.: “Coiled Tubing …Operations and Services, Part 4-Sand and Solids Washing” *World Oil* (Mar. 1992) 71-79.

6. Turner, R. G., Hubbard, M. G., and Dukler, A. E.: “Analysis and Prediction of Mininum Flow Rate for the Continuous Removal of Liquid from Gas Wells,” *J. Pet. Tech.* (Sept. 1969) 1475-1481.

Appendix F
Glossary

- A -

absolute pressure

The pressure measured from a vacuum. (0 psi absolute pressure = vacuum) Absolute pressure minus atmospheric pressure equals gauge pressure. Note that it is physically impossible to have a negative absolute pressure.

All pressures in Cerberus are gauge pressures.

annulus

The annulus is the area between the CT and the well tubulars.

application factor

Application factor serves as an additional safety factor. The actual value is multiplied by the application factor to produce the result. For example, if string fatigue has an application factor of 2, and if the calculated fatigue is 7%, it would appear as 14%.

archive

Indicates to Cerberus that a particular string, reel, or job is no longer used. Archived strings, reels, and jobs do not appear in the lists on the main manager screens unless desired.

axial force

Force applied along the length of the CT. Tension is positive. Compression is negative.

axial stress

The axial force at a given point in the CT divided by the cross-sectional area of the CT. The axial stress is part of the Von Mises stress.

azimuth

The azimuth in well survey data is the angle from North, in degrees, of the well axis at a given point. For example, a section of a well that goes East has an azimuth of 90 degrees.

- B -

BHA

In Cerberus, a bottom hole assembly (BHA) is called a tool string.

bias weld

A 45 degree weld made in the base material before the flat strip is rolled into tubing in the mill. Therefore, it is part of the manufacturing process and cannot be performing on existing tubing.

A bias weld is usually stronger than an orbital butt weld or manual butt weld but not as strong as parent tubing.

Bingham Plastic

The model for a fluid which does not flow until a specific yield point is reached. Cement and some drilling muds are often modeled as Bingham Plastic models.

- C -

compression

When a portion of the CT is pushed together along its axis, it is in compression.Compression is the opposite of tension.

continuous taper

Varying wall thickness of a section of CT linearly from one end of the section to the other.

critical buckling

The point where a section of CT begins to form a sinusoidal curve in the hole. Critical buckling does not mean that the CT cannot go further into the hole. Usually called sinusoidal buckling.

CT

An abbreviation for "coiled tubing".

- D -

default

The initial value set by the computer program if the user doesn't do anything. The program uses this value until the user changes it. Default values help speed data input and aid consistency for values that should always be the same.

derate

To manually lower the used life of a string to the derating factor. Used for welds and zones.

derating factor

The amount the used life of a weld or zone is reduced to. For example, a zone with a derating factor of 85% means that it has 85% of the life it would normally have.

depth

The distance from the zero depth datum at the top of the well to the point in question. Depth can be measured from the downhole end of a string, but depth is not the same as string position.

disconnect

Where a tool string connects to the CT. The disconnect should break before the CT. However, it should not break during normal operations.

download

To put strings, reels, jobs, wells, tool strings, fluids, projects, and other items to a file on removable media to share them with other people on other computers. The files must then be uploaded into Cerberus to make use of them. (Windows can not be used to simply copy the file, as Cerberus will not know the file exists.)

drawdown

The difference between the bottom hole pressure at the reservoir and the reservoir pressure. The change in pressure causes the fluid in a reservoir to flow into the well. In a killed well, there is no drawdown because the bottom hole pressure is equal to the reservoir pressure.

- E -

e-line capable

A work reel that has connections for an electrical cable installed inside the CT string.

effective wall thickness

The minimum wall thickness of a CT string minus the wall reduction. It represents the worst case for the actual wall thickness used in tubing life and strength calculations.

effective well

Used in force and hydraulics calculations. In general, the effective well follows the innermost tubulars (tubing, casing, liners, and open hole).

export

Survey data and job logs can be exported to text files which can then be imported to reuse them in different jobs, wells, or projects.

- F -

failure criterion

The point which Achilles uses as 100% used life for a string. Failure can be defined in two ways: crack initiation or fracture. In crack initiation, the beginnings of a crack appears on the CT. In fracture, the crack has propagated through the wall of the CT and the CT cannot hold pressure.

foam quality

The ratio of the volume of gas over the total volume of the fluid. A foam with a quality of 1 (one) is all gas. A foam with a quality of 0 (zero) is all liquid. Foam quality is dependent on temperature as well as pressure, as both affect the volume of gas.

freeboard

The distance from the top of the last wrap of coiled tubing to the outside of the flange of the reel. For safety reasons, the reel is never completely filled with coiled tubing. Some operators recommend a different freeboard for each coiled tubing diameter. The more freeboard, the less the reel capacity.

- G -

gauge pressure

The pressure indicated by a pressure gauge. (0 psi gauge pressure = atmospheric pressure.) Gauge pressure plus atmospheric pressure equals absolute pressure. Note that it is possible to have a negative gauge pressure. All pressures in Cerberus are gauge pressures.

- H -

helical buckling

The point where a section of CT begins to form a helix in the well casing. In a vertical well, helical buckling begins as soon as any compressive load is applied. The helix introduces additional bending stresses and friction, which can lead to lockup.

helical buckling load

The amount of force required to reach helical buckling.

hoop stress

The stress around the circumference of the CT due to inner and outer pressure. The hoop stress is part of the Von Mises stress.

- I -

import

Survey data and job logs can be imported from a text file format for use or reuse. The opposite of export.

inclination

In well survey data, the angle, in degrees, between the well axis and the vertical axis at a given depth. A vertical section of well has an inclination of 0 degrees. A completely horizontal section of well has an inclination of 90 degrees.

- J -

JP

An abbreviation for "jointed pipe".

- K -

- L -

lockup depth

The depth, if any, at which the increased drag due to helical buckling causes the downhole end of the tubing to stop moving into the well, even though CT may still be entering the well at the surface. If this occurs, the bottom of the well cannot be reached and the job cannot be performed with the existing configuration.

- M -

manual butt weld

Performed by hand to join two lengths of tubing. It is the typical method used to repair tubing in the field. The main characteristic of manual butt welds is variability from welder to welder, and even weld to weld. However, a good manual weld can be superior to an orbital butt weld.

measured depth

The current depth of the CT following the trajectory of the well as indicated by the depth counter at the surface. The actual depth may be slightly more due to buckling or stretch and thermal elongation. Note that measured depth is different from true vertical depth (TVD).

minimum wall thickness

The smallest allowable thickness for a given nominal wall thickness, according to the manufacturer's specifications. Although the actual wall thickness will, in most cases, be greater, for critical calculation purposes, the minimum wall thickness is used to ensure that a worst-case scenario is considered.

Multiphase fluid

A fluid, generally a liquid, comprising more than one phase, such as water- or oil-based liquids, and gas. Multiphase fluid flow is the combined flow of these different-phase fluids. Multiphase flow is a complex factor that is important in understanding hydraulics in both oil and gas wells.

- N -

Newtonian fluid

The model for fluids that flow in a linear manner. Few oil field fluids are truly Newtonian, but the Newtonian model is the best known and easiest to understand. Fresh water can be modeled as a Newtonian fluid.

nominal wall thickness

The target wall thickness sold by the manufacturer and commonly referred to by users. The actual wall size may vary within the plus and minus specifications quoted by the manufacturer.

- O -

orbital butt weld

Performed using an automated welder to join two lengths of tubing. The resulting weld is typically more consistent than a manual butt weld, but is not necessarily better. Orbital welders are expensive and are not often encountered in the field.

- P -

POOH

An abbreviation for "pulling out of hole".

position

The distance measured from the reel core end of a string to the point in question. The reel core end is at position zero. Position is not related to depth, although depth is measured from the free end of a string.

Power Law Model

The model for fluids which do not flow in a linear manner. Many oil field fluids, including water-based muds, gels, and hydrocarbons, are modeled using the Power Law model.

- Q -

- R -

radial stress

The stress through the CT wall due to inner and outer pressure. The radial stress is part of the Von Mises stress.

reel back tension

The tension maintained at the reel to ensure that the tubing on the reel does not uncoil or become slack. Reel back tension only affects the surface weight calculation, not the downhole forces, since the tubing is held at the injector.

RIH

An abbreviation for "running in hole".

RTKB

Rotary Table Kelly Bushing. The depth reference typically used by conventional drilling rigs when they drill a well.

- S -

section

A CT string is divided into sections which have different physical properties, such as different wall sizes.

segment

A CT string is divided into small segments of equal length (usually 10 feet). Fatigue is calculated for each segment.

segment length

The length of a segment, such as 10 feet. The total length of a string must be a multiple of the segment length.

shipping spool

Used to hold a CT string for shipping purposes only. A shipping reel is not used for a job. Instead, a work reel is used for a job.

sinusoidal buckling

The CT snaking back and forth in the form of a sine wave. Also called snake buckling or critical buckling. Sinusoidal buckling is not critical and does not affect tubing forces calculations.

sinusoidal buckling load

The load at which the CT begins sinusoidal buckling.

stripper friction

The frictional force caused by the compression of the stripper packing elements on the tubing. This friction always acts in the opposite direction of the tubing movement. Although variable, it is usually assumed to lie in the range of 300 to 1,000 pounds of force.

surface weight

The weight measured at the surface as the CT is run into and out of the well. This weight should stay below the yield strength of the CT material or breaking strength of the cable, subject to a safety factor, such as 80%.

- T -

target depth

The particular depth at which most hydraulics calculations are performed.

tension

When a portion of CT is being pulled, it is in tension. Tension is the opposite of compression.

tool

A single component in a tool string, such as a connector, nozzle, or packer.

tool string

A bottom hole assembly (BHA). It is composed of various tools.

tortuosity

The "spiraling" or "corkscrew" effect that occurs when wells are being drilled.

trip

One cycle of the string running into and out of the hole.

true vertical depth (TVD)

The absolute depth from the surface, not following the trajectory of the well.

- U -

upload

The process of importing information for a string, reel, job, well, tool string, fluid, project, or other item from removable media. This must be done in order for Cerberus to use such existing data. Further, the item must have been downloaded onto the removable media.

used life

Used life of a work string comes from accumulated fatigue and any corrosion. The used life of a work string is represented by a percentage. A new string has 0% used life. A work string should be retired or cut before it reaches too high a used life.

- V -

Von Mises stress

The Von Mises stress (σ_{vm}) is a common method for describing the yielding of steel under combined states of stress. It includes the influences of axial stress (σ_a), radial stress (σ_r), and hoop stress (σ_h).

$$\sigma_{vm} = \sqrt{\frac{1}{2}\left[(\sigma_a - \sigma_r)^2 - (\sigma_a - \sigma_h)^2 - (\sigma_r - \sigma_h)^2\right]}$$

- W -

wall reduction

The change in wall thickness due to exposure to service conditions such as acid jobs and sand abrasion. In Cerberus, wall reduction is a function of the jobs the string has performed and is set for each job in the Job Type screen of Job Manager.

work reel

Used for an actual job, as opposed to a shipping spool, which is only used for shipping or storage.

work string

A CT string which is actively used in coiled tubing operations. When a work string accumulates too much fatigue, it may be retired and used for other purposes.

- X -

- Y -

Young's Modulus

The modulus of elasticity of the material. Young's Modulus is a measurement of how 'springy' a material is, or how much a material can bend and still return to its original shape. Young's Modulus for steel is 30,000,000 psi.

- Z -

zero depth datum

The point from which depth is measured at the well site (depth = 0). This may be at the Master Valve, ground level, RTKB, at the reel, or any other user-specified position.

zone

Used to flag portions of a CT string and to manually derate fatigue life.